

Terramodel[®] User Guide



Version 7
Part Number 0321-0800
Revision A
April 2002

Corporate Office

Trimble Navigation Limited
Engineering and Construction
Division
5475 Kellenburger Road
Dayton, Ohio 45424-1099
U.S.A.

Copyright and Trademarks

© 2002, *Trimble Navigation* Limited.
All rights reserved. Printed in the United States of *America*. *Printed on recycled* paper. The Globe & Triangle logo, Trimble, and Terramodel are trademarks of Trimble Navigation Limited. The Sextant logo with Trimble is a trademark(s) of Trimble Navigation Limited, registered in the United States Patent and Trademark Office. All other trademarks are the property of their respective owners.

Release Notice

This is the April 2002 release (Revision A) of the Terramodel User Guide. It applies to version 10.1 of the Terramodel® software.

The following limited warranties give you specific legal rights. You may have others, which vary from state/jurisdiction to state/jurisdiction.

Software and Firmware License, Limited Warranty

This Trimble software and/or firmware product (the “Software”) is licensed and not sold. Its use is governed by the provisions of the applicable End User License Agreement (“EULA”), if any, included with the Software. In the absence of a separate EULA included with the Software providing different limited warranty terms, exclusions,

and limitations, the following terms and conditions shall apply.

Trimble warrants that this Trimble Software product will substantially conform to Trimble’s applicable published specifications for the Software for a period of ninety (90) days, starting from the date of delivery.

Software and Firmware License, Limited Warranty

This Trimble software and/or firmware product (the “Software”) is licensed and not sold. Its use is governed by the provisions of the applicable End User License Agreement (“EULA”), if any, included with the Software. In the absence of a separate EULA included with the Software providing different limited warranty terms, exclusions, and limitations, the following terms and conditions shall apply. Trimble warrants that this Trimble Software product will substantially conform to Trimble’s applicable published specifications for the Software for a period of ninety (90) days, starting from the date of delivery.

Warranty Remedies

Trimble's sole liability and your exclusive remedy under the warranties set forth above shall be, at Trimble’s option, to repair or replace any Product or Software that fails to conform to such warranty (“Nonconforming Product”), or refund the purchase price paid by you for any such Nonconforming Product, upon your return of any Nonconforming Product to Trimble in accordance with Trimble’s standard return material authorization procedures.

Warranty Exclusions and Disclaimer

These warranties shall be applied only in the event and to the extent that: (i) the Products and Software are properly and correctly installed, configured, interfaced, maintained, stored, and operated in accordance with Trimble's relevant operator's manual and specifications, and; (ii) the Products and Software are not modified or misused. The preceding warranties shall not apply to, and Trimble shall not be responsible for defects or performance problems resulting from (i) the combination or utilization of the Product or Software with products, information, data, systems or devices not made, supplied or specified by Trimble; (ii) the operation of the Product or Software under any specification other than, or in addition to, Trimble's standard specifications for its products; (iii) the unauthorized modification or use of the Product or Software; (iv) damage caused by accident, lightning or other electrical discharge, fresh or salt water immersion or spray; or (v) normal wear and tear on consumable parts (e.g., batteries).

THE WARRANTIES ABOVE STATE TRIMBLE'S ENTIRE LIABILITY, AND YOUR EXCLUSIVE REMEDIES, RELATING TO PERFORMANCE OF THE PRODUCTS AND SOFTWARE. EXCEPT AS OTHERWISE EXPRESSLY PROVIDED HEREIN, THE PRODUCTS, SOFTWARE, AND ACCOMPANYING DOCUMENTATION AND MATERIALS ARE PROVIDED "AS-IS" AND WITHOUT EXPRESS OR IMPLIED WARRANTY OF ANY KIND BY EITHER TRIMBLE NAVIGATION LIMITED OR ANYONE WHO HAS BEEN INVOLVED IN ITS CREATION, PRODUCTION, INSTALLATION, OR DISTRIBUTION, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, TITLE, AND NONINFRINGEMENT. THE STATED

EXPRESS WARRANTIES ARE IN LIEU OF ALL OBLIGATIONS OR LIABILITIES ON THE PART OF TRIMBLE ARISING OUT OF, OR IN CONNECTION WITH, ANY PRODUCTS OR SOFTWARE. SOME STATES AND JURISDICTIONS DO NOT ALLOW LIMITATIONS ON DURATION OR THE EXCLUSION OF AN IMPLIED WARRANTY, SO THE ABOVE LIMITATION MAY NOT APPLY TO YOU. TRIMBLE NAVIGATION LIMITED IS NOT RESPONSIBLE FOR THE OPERATION OR FAILURE OF OPERATION OF GPS SATELLITES OR THE AVAILABILITY OF GPS SATELLITE SIGNALS.

Limitation of Liability

TRIMBLE'S ENTIRE LIABILITY UNDER ANY PROVISION HEREIN SHALL BE LIMITED TO THE GREATER OF THE AMOUNT PAID BY YOU FOR THE PRODUCT OR SOFTWARE LICENSE OR U.S.\$25.00. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL TRIMBLE OR ITS SUPPLIERS BE LIABLE FOR ANY INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER UNDER ANY CIRCUMSTANCE OR LEGAL THEORY RELATING IN ANY WAY TO THE PRODUCTS, SOFTWARE, AND ACCOMPANYING DOCUMENTATION AND MATERIALS, (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, OR ANY OTHER PECUNIARY LOSS), REGARDLESS OF WHETHER TRIMBLE HAS BEEN ADVISED OF THE POSSIBILITY OF ANY SUCH LOSS AND REGARDLESS OF THE COURSE OF DEALING WHICH DEVELOPS OR HAS DEVELOPED BETWEEN YOU AND TRIMBLE. BECAUSE SOME STATES AND JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL OR INCIDENTAL DAMAGES, THE ABOVE LIMITATION MAY NOT APPLY TO YOU.

Contents

ONE: INTRODUCTION OVERVIEW & INSTALLATION	21
CONVENTIONS AND DEFINITIONS	21
<i>Mouse Conventions</i>	22
HOW TO USE TERRAMODEL DOCUMENTATION	23
<i>User Guide</i>	23
<i>Status bar</i>	23
<i>On-Line Help System</i>	23
<i>Instant Expert On-line Guides</i>	24
<i>Telephone support and Training</i>	24
<i>www.trimble.com</i>	24
<i>TML On-Line help</i>	25
OVERVIEW OF TERRAMODEL	25
TERRAMODEL MODULES	25
FIELD DATA MODULE COMMANDS	27
COMMANDS SECURED TO MODULES	28
ON-LINE HELP	31
TELEPHONE TECHNICAL SUPPORT	31
TERRAMODEL TRAINING	32
MINIMUM REQUIREMENTS	33

INSTALLING TERRAMODEL	33
<i>To install Terramodel:</i>	34
<i>Hardware Lock Drivers</i>	35
<i>To install the hardware lock drivers manually:</i>	36
UNINSTALLING TERRAMODEL	36
THE TERRAMODEL SEARCH PATH	36
<i>Terramodel Folders</i>	36
<i>The TSP Command</i>	37
HOW TO USE CUSTOMIZATION FOLDERS	38
<i>Create the customization folder</i>	39
<i>Edit TMODWIN.INI</i>	40
<i>Set the TMODWIN Environment Variable</i>	41
THE TERRAMODEL HARDWARE LOCK	42
TERRAMODEL OPENING STATE.....	43
<i>Terramodel Without a Hardware Lock</i>	43
<i>Terramodel with a Hardware Lock</i>	43
CONFIGURE THESE CRITICAL SETTINGS	44
<i>System Settings</i>	44
<i>Project Settings</i>	45
<i>Project Settings that Reflect Unit Settings</i>	46
CONFIGURE THESE SETTINGS FOR LOCALIZATION.....	47
<i>X Y or Y X</i>	47
<i>Station and Chainage</i>	47
<i>Rise/Run or Run/Rise</i>	48
<i>North, South, East Azimuths or Bearings</i>	48
<i>Extended ASCII Characters</i>	48
SETUP	48
SYSTEM CONFIGURATION.....	49
THE TERRAMODEL INITIALIZATION FILES	53
TWO: THE TERRAMODEL DISPLAY	55
WINDOWS BASICS	56
<i>Views (Windows)</i>	56
<i>Control Menus</i>	58
<i>Dialog Boxes</i>	58
<i>Basic Windows Controls</i>	60
<i>Key Combinations</i>	61
MOUSE POINTER SHAPES	63
COMMON BUTTON CONTROLS	63
THE MENU BAR	64
<i>Menu Configuration Options</i>	66

THE STATUS BAR.....	67
<i>Status Bar Sections</i>	67
THE TOOLBAR.....	68
<i>Toolbar Controls</i>	68
<i>The Default Toolbar</i>	69
<i>Toolbar Toggles</i>	70
<i>Layer Setup Dialog Box</i>	71
<i>Layer Selection Control</i>	72
<i>Layer Selection Dialog Box</i>	72
<i>Layer and Point Color Controls</i>	72
<i>Linetype Selection Control</i>	73
<i>Linetype Dialog Box</i>	73
THE COORDINATE SCROLL.....	74
<i>Coordinate Scroll Options</i>	74
THE COMMAND BAR.....	76
THE MESSAGE SCROLL.....	77
THE TERRAMODEL REPORT EDITOR.....	78
THREE: HOW TO CREATE, SAVE & RETRIEVE PROJECTS	79
CREATING A NEW PROJECT.....	79
BUILDING A PROJECT FILE.....	81
SAVING PROJECT FILES.....	85
CLOSING PROJECT FILES.....	86
RETRIEVING PROJECT FILES.....	86
CREATING A PROTOTYPE PROJECT FILE.....	88
FOUR: UNDERSTANDING TERRAMODEL OBJECTS	91
TERRAMODEL OBJECTS.....	91
UNDERSTANDING POINTS.....	93
<i>How to Create Points</i>	93
<i>How to Select Points</i>	95
UNDERSTANDING SETS.....	96
<i>What Sets are Made Of</i>	96
<i>Open, Closed and Joined Sets</i>	96
<i>Curved Set segments</i>	97
<i>What Sets are Used For</i>	98
<i>How to Create Sets</i>	100
<i>How to Select Sets</i>	101
UNDERSTANDING POLYLINES.....	102
<i>What a Polyline Consists Of</i>	102

<i>Types of Polylines You Can Create</i>	102
<i>What Polylines are Used For</i>	103
<i>How to Create Polylines</i>	104
<i>How to Select Polylines</i>	105
UNDERSTANDING TEXT	106
<i>Text Objects</i>	106
<i>Attribute Text</i>	110
<i>How to Create and Edit Text</i>	111
<i>How to Select Text</i>	112
UNDERSTANDING BLOCKS	113
<i>What Blocks are Made Of</i>	113
<i>What Blocks are Used For</i>	113
<i>How to Create Blocks</i>	114
<i>Exploding Blocks</i>	114
UNDERSTANDING HORIZONTAL AND VERTICAL ALIGNMENTS	115
<i>Horizontal Alignments</i>	115
<i>Vertical Alignments</i>	116
<i>How HALs and VALs are Used Together</i>	116
<i>How to Create HALS and VALS</i>	117
UNDERSTANDING DYNAVIEWS	119
<i>How to Create Dynaviews</i>	119
<i>How Objects are Displayed through a Dynaview</i>	120
<i>Dynaviews and Object Referencing</i>	121
<i>Changing a Dynaview</i>	122
<i>Advantages of Using Dynaviews</i>	122
<i>Commands Associated with Dynaviews</i>	123
<i>How to Select Dynaviews</i>	123
UNDERSTANDING ATTRIBUTES	124
REFERENCING OBJECTS TO OTHER OBJECTS	124
<i>Active Alignments</i>	125
FIVE: UNDERSTANDING TERRAMODEL CAD	127
VIEW MODES	127
<i>Plan View</i>	128
<i>Profile View</i>	128
<i>Sheet View</i>	128
<i>Xsection View</i>	129
<i>Superelevation View</i>	129
<i>Views 6–8</i>	129
<i>View mode properties</i>	129
WORKING WITH LAYERS	130
<i>Choosing Layer Names</i>	130

<i>Assigning Layer Colors</i>	131
<i>Layer Settings Dialog Box</i>	132
<i>Layer Offset Dialog Box</i>	136
<i>New Layer Dialog Box</i>	136
<i>Layer List Selection Dialog Box</i>	137
<i>Current Layers and the Layer Selection Control</i>	137
<i>Layer Selection Dialog Box</i>	138
<i>Moving Objects to Other Layers</i>	138
WORKING WITH LAYER LISTS.....	139
<i>Layer List Settings Dialog Box</i>	140
<i>New Layer List Dialog Box</i>	142
<i>Rename Layer List Dialog Box</i>	142
<i>Assigning a Layer List to a Dynaview</i>	142
<i>Controlling Background Object Colors</i>	143
<i>Controlling the Pen Selection for Background Objects</i>	144
WORKING WITH COLORS	145
<i>Color Selection Dialog Box</i>	145
<i>Creating Your Own Palettes and Color Maps</i>	146
<i>Mapping Colors to Pens</i>	146
WORKING WITH LINETYPES.....	147
<i>Loading a Linetype File</i>	147
<i>Changing the Linetype of the Current Layer</i>	147
<i>Linetype Dialog Box</i>	148
<i>Changing the Linetype of Selected Objects</i>	148
<i>Working with Linetypes</i>	148
CREATING CUSTOM LINETYPES.....	149
<i>The Four Line Types</i>	149
<i>The .LIN File Format</i>	150
<i>Using Symbols in Linetypes</i>	151
<i>Example Linetype File</i>	152
WORKING WITH TEXT AND TEXT STYLES	156
<i>Text Fonts</i>	156
<i>Font Shape files</i>	156
<i>Creating Fonts from Shapes</i>	157
<i>Superscripts and Subscripts</i>	157
<i>Creating Shape Files From SHX Files</i>	158
<i>Notice of Restricted Use</i>	158
<i>Text Leader Lines, Text Borders and Callouts</i>	159
<i>Mirrored Text</i>	159
<i>Smart Text—Embedded Attribute Text (EAT) Codes</i>	160
<i>Text from External Files</i>	167
<i>Drafting Text</i>	167
<i>Text Styles</i>	170

<i>Text Style Settings Dialog Box</i>	170
EXTENDED ASCII CHARACTERS	173
WORKING WITH SYMBOLS	173
WORKING WITH HATCH PATTERNS.....	175
<i>Hatching with the Pattern Option</i>	175
<i>Hatch Pattern Dialog Box</i>	176
<i>Hatching with the User Option</i>	176
CREATING CUSTOM HATCH PATTERNS	177
<i>Hatch Pattern Example</i>	178
WORKING WITH DYNAVIEWS	179
ADDITIONAL ADD-INS	181
<i>Title Blocks</i>	181
<i>North Arrow Blocks</i>	182
<i>Graphic Scale Blocks</i>	183
<i>Linetypes</i>	185
<i>Fonts</i>	197
SAMPLE BLOCKS	200
SIGN BLOCKS.....	200
SIX: USING TERRAMODEL CUSTOM CONTROLS.....	209
CAD CONTROLS	210
<i>Color Control</i>	210
<i>Edit Control</i>	210
<i>Layer Control</i>	211
<i>Linetype Control</i>	211
<i>Hatch (Pattern) Control</i>	212
<i>Text Size Control</i>	213
<i>Name Control</i>	213
LOCATE CONTROLS	214
<i>Locate (Object)</i>	214
<i>Locate (Element)</i>	215
<i>Locate (Browse)</i>	215
<i>Left/Right Control</i>	215
NUMBER CONTROLS	216
<i>Point Number Control</i>	216
<i>Integer Number Control</i>	216
<i>Real Number Control</i>	216
COORDINATE GEOMETRY CONTROLS.....	218
<i>Angle Control</i>	218
<i>Bearing Control</i>	219
<i>Point (Coordinate) Control</i>	221

<i>Distance Control</i>	223
<i>Elevation Control</i>	224
<i>Hide and Pick Control</i>	224
<i>Offset Control</i>	224
<i>Station Control</i>	225
<i>Vertical (Angle) Control</i>	226
SELECT CONTROLS	228
<i>Select by Color Dialog Box</i>	234
<i>Select by Layer(s) Dialog Box</i>	235
<i>Select by Linetype Dialog Box</i>	236
<i>Select by Name Dialog Box</i>	237
<i>NearLine Command Bar</i>	237
<i>Select by Point Number Dialog Box</i>	238
<i>Select by Type Dialog Box</i>	239
<i>Select by View Dialog Box</i>	239
<i>Select By Window Command Bar</i>	240
<i>Select Expressions</i>	240
SEARCHING FOR OBJECTS	245
OBJECT SNAP MODES.....	247
<i>Point Snap Modes</i>	247
<i>Bearing Snap Modes</i>	254
<i>Distance Snap Modes</i>	256
<i>Vertical Snap Modes</i>	257
<i>Angle Snap Modes</i>	258
<i>Elevation Snap Modes</i>	259
<i>Station Snap Modes</i>	260
<i>Offset Snap Modes</i>	261
RUNNING SNAP MODES.....	263
<i>Running Snap Modes Dialog Box</i>	263
SEVEN: CREATING A DIGITAL TERRAIN MODEL.....	263
WHAT IS A DTM?.....	263
LINKS AND A DTM	264
<i>Link Settings Dialog Box</i>	264
<i>Changing the Links with DtmEdge</i>	268
ADDING BREAKLINES TO A DTM.....	269
SPLINE TYPES.....	272
CONTROLLING THE DTM EDGE	273
MULTILAYER DTMS	274

EIGHT: GETTING DIGITAL DATA INTO/OUT OF TERRAMODEL	275
INTRODUCTION	275
<i>Terramodel Download Options</i>	276
<i>Import File Format Options</i>	277
<i>Download/Import Data Format Template (a.k.a. .lgr or logger file)</i>	278
<i>Export File Format Options</i>	279
<i>Terramodel Upload Options</i>	279
THE IMPORT/EXPORT COMMANDS	280
<i>Naming Scripts</i>	281
<i>Import or Export Using the File Menu</i>	281
<i>Import or Export Using the Terramodel Command Line</i>	282
SCRIPT MANAGER.....	285
<i>Creating Scripts</i>	286
<i>Editing Scripts</i>	289
<i>Editing a Script – an Example</i>	290
<i>Deleting Scripts</i>	291
CREATING SCRIPTS VS. RUNNING SCRIPTS.....	292
NINE: RAW DATA EDITOR	293
THE RAW DATA EDITOR	294
<i>RDE Features</i>	294
<i>Field Data Reduction – Fast Start</i>	296
RDE DIALOG BOX.....	299
<i>Starting the RDE</i>	299
<i>RDE Menu Bar</i>	300
<i>Survey Job Control</i>	302
<i>Display Pane</i>	302
<i>Message Pane</i>	308
<i>Detail Pane</i>	310
<i>Settings Menu Options</i>	311
<i>Report Options</i>	317
TERRAMODEL GRAPHICS INTERFACE.....	317
<i>Point objects</i>	317
<i>Layer Assignment of Terramodel Project Objects</i>	318
<i>Importing Data and the RDE</i>	319
<i>Terramodel Commands and the RDE</i>	321
SURVEYING BASICS WITH THE RDE.....	322
<i>Description of an Observation</i>	322
<i>Types of Observations</i>	323
<i>Description of a Traverse</i>	325
SURVEYING ERRORS AND TOLERANCES	326

<i>Precision, Accuracy & Significant Figures</i>	326
<i>Errors</i>	327
LEAST SQUARES	328
<i>Unique Solution – the Benefit of Least Squares</i>	328
SURVEY TECHNIQUE SUGGESTIONS	329
RDE – CALCULATION METHODS	331
<i>Introduction</i>	331
<i>Step 1: Reduction of Observations</i>	332
<i>Step 2: Calculate Provisional Coordinates</i>	335
<i>Step 3: Calculate Final Coordinates with Least Squares</i>	337
<i>Step 4: Calculate Sideshots</i>	339
KEY NAMES AND DATA TAGS	339
TEN: REFERENCE FILES	341
TERMINOLOGY	341
<i>Master Project</i>	341
<i>Reference Project</i>	342
<i>Project Reference Prefix</i>	342
<i>Active, and Background Projects</i>	342
<i>Editable State</i>	343
<i>Read-Only State</i>	343
<i>Read-Write Project</i>	343
<i>Locked Project</i>	343
PROPERTIES OF REFERENCE FILES	345
<i>The Reference File Manager</i>	345
<i>Project Data Access Control</i>	347
<i>Displayed Content</i>	349
<i>Updating Reference File Content</i>	349
<i>Copying Data Between Projects</i>	349
<i>General Conditions</i>	351
ELEVEN: DESIGNING WITH ROAD, PIPE, & HYDRO	353
HOW TO USE THE ROADWAY COMMANDS	354
<i>Understanding Roadway Concepts and Terminology</i>	354
CREATING A ROADWAY DESIGN	363
USING ROAD JOB STATIONING (ROADJOB)	365
DESIGNING A SINGLE ROADWAY	366
DESIGNING MULTIPLE ROADWAYS	368
MASS HAUL ANALYSIS	370
PREPARING A PROJECT TO DO A MASS HAUL ANALYSIS	371
<i>Defining Materials Available at the Job Site</i>	371

<i>Setup Phases for the Road Job</i>	374
<i>Set up Your Significant Points</i>	374
<i>Establishing Subgrades</i>	377
<i>Define Hauling Properties for Road Job Materials</i>	379
USING THE MASS HAUL DIAGRAM	382
<i>Mass Haul Diagram/Actions Menu</i>	385
<i>Mass Haul Diagram/Display Menu</i>	386
<i>Mass Haul Diagram Grid Settings</i>	389
<i>Import Cut/Fill Volumes to use in MassHaul Analysis (MhImport)</i>	392
HOW TO USE THE PIPE COMMANDS.....	393
<i>Drainage Concepts and Terminology</i>	393
FLOW RATE CALCULATION METHODS	395
<i>Q Summation</i>	395
<i>Rational Method (Q=CIA)</i>	395
<i>None</i>	397
HOW TO ASSIGN PIPE SIZES	398
DESIGNING A SEWER SYSTEM	401
ANALYZING AN EXISTING SEWER SYSTEM.....	404
HOW TO USE THE HYDRO COMMANDS.....	406
<i>Understanding Hydrology Concepts and Terms</i>	406
<i>Hydrograph Calculation Methods</i>	407
DESIGNING A SIMPLE BASIN.....	411
DESIGNING MULTIPLE BASINS	417
CUSTOMIZING THE HYDROLOGY MODULE USING TML	418
HYDROLOGY MODULE REFERENCES.....	419
TWELVE: AUTOMATED SHEET ASSEMBLY AND PRODUCTION.....	419
GETTING STARTED.....	419
<i>Using the Plan Set Commands</i>	419
<i>Using the ASAP commands</i>	421
WORKING WITH PLAN SETS	422
<i>Plan Set Concepts and Terminology</i>	422
<i>Plan Set Configuration Settings</i>	424
<i>Supported Drawing Configurations</i>	425
<i>Creating an Index of Drawings</i>	426
<i>Automated Sheet Assembly</i>	427
WORKING WITH SHEETS.....	428
<i>Creating a Base Sheet</i>	428
<i>Creating a Master Sheet</i>	430
<i>Creating Plotboxes</i>	431
<i>Creating Drawing Sheets</i>	438

<i>Recreating Drawing Sheets</i>	441
<i>Creating Blank Sheets</i>	442
<i>Selecting Sheets</i>	443
<i>Placing the Sheet Insertion Point Marker</i>	443
<i>Adjusting Drawing Sheet Components</i>	444
<i>Combining Sheets</i>	444
<i>Reordering Sheets</i>	445
<i>Deleting a Sheet</i>	445
<i>Viewing Individual Sheets and Sheet Information</i>	446
<i>Plotting Sheets</i>	447
USING THE PROJECT DATA EDITOR	447
EDITING PROJECT DATA	449
<i>General Information Dialog Box</i>	451
<i>Project Location Dialog Box</i>	452
<i>Client Data Dialog Box</i>	453
<i>Staff Data Dialog Box</i>	454
LINKING TEXT OBJECTS TO PROJECT DATA	455
CUSTOMIZING THE PROJECT DATA CONTENT	456
THIRTEEN: USING COORDCON	457
TERRAMODEL COORDINATE CONVERSION ROUTINES	457
<i>Uses for Coordcon in Terramodel</i>	458
<i>Map Projections and Coordinate Transformations</i>	459
THE LATITUDE AND LONGITUDE COORDINATE SYSTEM	463
CONVERGENCE OF MERIDIANS	466
GEODETIC CONTROL NETWORKS AND EARTH DATUMS	468
THE EARTH AS AN ELLIPSOID	470
<i>Ellipsoid Equatorial and Polar Radii</i>	474
<i>Ellipsoid Flattening</i>	474
<i>Ellipsoid Eccentricity</i>	475
UNITED STATES ELLIPSOIDS AND DATUMS	476
<i>The North American Datum of 1927 (NAD-27)</i>	476
<i>North American Datum of 1983 (NAD-83)</i>	477
MAP PROJECTIONS	479
<i>Azimuthal Map projection</i>	479
<i>Cylindrical (Mercator) Map Projection</i>	481
<i>Conic Map Projection</i>	483
<i>Transverse Cylindrical Projection</i>	485
<i>The Transverse Mercator Projection</i>	493
<i>The Universal Transverse Mercator Projection</i>	497
<i>The Lambert Conformal Conic Projection</i>	499

<i>The Hotine Oblique Mercator Projection</i>	502
STATE PLANE COORDINATE SYSTEMS.....	504
HOW TO CONVERT COORDINATES USING COORDCON.....	506
<i>Before Using CoordCon</i>	506
<i>Search Coordinate Systems Dialog Box</i>	508
<i>Defining a Coordinate System</i>	510
<i>Defining an Ellipsoid</i>	512
<i>Defining a Datum</i>	514
GLOSSARY OF GEODETIC TERMS.....	517
FOURTEEN: CREATING REPORTS	527
<i>The P3Pad Report Editor</i>	527
CREATING A REPORT.....	528
SAVING A REPORT.....	528
<i>Print to File</i>	528
<i>Saving a Formatted Report</i>	529
EDITING REPORT TEXT.....	530
<i>Adding Text</i>	530
<i>Selecting Text</i>	530
<i>Deleting Text</i>	530
<i>Copying Text</i>	531
<i>Moving Text</i>	531
<i>Undo</i>	531
FINDING AND REPLACING TEXT.....	532
<i>Find and Find Next</i>	532
<i>Replacing Text</i>	533
FORMATTING THE REPORT.....	534
<i>Text Formatting with the Font Dialog Box</i>	534
<i>Text Formatting from the Format Bar</i>	535
<i>Setting Paragraph Indents</i>	537
<i>Setting and Clearing Tabs</i>	537
<i>Setting Page Layout</i>	538
INSERTING OBJECTS INTO THE REPORT.....	540
<i>Inserting the Date and Time</i>	540
<i>Inserting Objects</i>	540
<i>Inserting with Paste Special</i>	545
PRINTING THE REPORT.....	546
<i>Print Preview</i>	546
<i>Printing the Report</i>	547

FIFTEEN: TERRAMODEL VISUALIZER	549
INTRODUCTION	549
<i>Terramodel Visualizer Documentation</i>	550
<i>Installing Terramodel Visualizer</i>	550
STARTING AND USING TERRAMODEL VISUALIZER	551
<i>Terminology</i>	552
<i>Application Ideas for Terramodel Visualizer</i>	552
<i>Using Terramodel Project Files in Terramodel Visualizer</i>	554
<i>Learning How to Use Terramodel Visualizer</i>	554
<i>Example of Following a Recorded Path</i>	555
<i>Example of using the rotating mode</i>	556
<i>Example of using the flying mode</i>	556
<i>Example of saving an image</i>	557
<i>Explore other example projects</i>	558
SIXTEEN: HOW TO PLOT INTERRAMODEL.....	559
PREPARING TO PLOT FROM TERRAMODEL.....	559
<i>Configuring Your Plotter</i>	560
<i>Using Pen Carousels</i>	566
<i>Setting Plot Resolutions</i>	569
<i>Mapping Colors to Pens</i>	570
<i>Advanced plotting features</i>	575
PLOTTING A PROJECT FILE.....	577
<i>Plotting to a Device or a File</i>	577
SEVENTEEN: DIGITIZER SETUP	581
<i>Get Your Digitizer Ready</i>	581
CONFIGURING YOUR DIGITIZER	582
ORIENTING DIGITIZER MENUS AND SCREEN AREAS	584
<i>Creating a New Menu</i>	586
<i>Creating a New Screen Area</i>	588
ORIENTING THE DIGITIZER TO YOUR PROJECT.....	591
WINTAB TABLET DRIVER.....	593
EIGHTEEN: CUSTOMIZINGTERRAMODEL.....	595
USING THE TOOLBOX.....	596
<i>Getting Started</i>	596
USING WORKSPACES.....	597
<i>The Default Workspace</i>	597
<i>Opening a Workspace</i>	597
<i>Creating a New Workspace</i>	598

<i>Saving a Workspace</i>	598
<i>The Library Workspace</i>	599
USING TOOLBOXES	600
<i>Parts of a Toolbox</i>	600
<i>Using the Toolbox Control Menu</i>	601
<i>Manipulating Toolboxes on the Screen</i>	601
<i>The Toolboxes Dialog Box</i>	602
USING COMMAND BUTTONS.....	607
<i>Executing a Command</i>	608
<i>Viewing the Command Button Title</i>	608
<i>Changing the Contents of Your Toolboxes</i>	608
<i>The Edit Command Button Dialog Box</i>	610
<i>Creating a New Command Button</i>	612
<i>Viewing Notes</i>	612
<i>Using the Command Toolbox</i>	613
<i>Using the Icon Toolbox</i>	613
USING CASCADE BUTTONS.....	613
<i>Creating a Cascade Button</i>	613
<i>Using a Cascade Button</i>	614
<i>Tearing Off a Toolbox</i>	614
<i>Unlinking a Cascade Button</i>	615
TOOLBOX COMMAND SYNTAX.....	615
FLOATING COMMAND BUTTONS.....	615
<i>Floating Another Button to the Top Position</i>	615
<i>Keeping the New Button on Top</i>	616
USING THE TOOLBOX BITMAP EDITOR.....	616
<i>Opening the Bitmap Editor</i>	617
<i>The Bitmap Editor</i>	618
<i>Creating a New Bitmap Icon from Scratch</i>	621
<i>Creating a New Bitmap Icon Using an Existing Bitmap</i>	622
CUSTOMIZING THE TERRAMODEL TOOLBAR.....	624
<i>The Toolbar</i>	624
CUSTOMIZING SPECIAL FUNCTION KEYS	624
USING THE TERRAMODEL MACRO LANGUAGE.....	627
<i>Where to Learn About Programming with TMLs</i>	627
TML BASICS	628
INDEX	i
APPENDIX A: TV TREES, SHRUBS, AND EXAMPLES	xi
INTRODUCTION	xi

Trees, Shrubs and Geotypical Texturesxi
Graphics of example projects..... xii
Tree and Shrub File Names (see page 658.)xiv
Tree and Shrub File Names (see page 660.) xv
Tree and Shrub File Names (see page 662.)xvi
Tree and Shrub File Names (see page 664.)xvii
TREE AND SHRUB PICTURE 1 xix
TREE AND SHRUB PICTURE 2 xxi
TREE AND SHRUB PICTURE 3xxiii
TREE AND SHRUB PICTURE 4xxv
EXISTING GROUND PICTURExxvii
PIPELINE PICTURE..... xxix
SUBDIVISION LOTS PICTURE xxxi

Chapter One

Introduction Overview & Installation

Conventions and Definitions

We use special typographic conventions and mouse terminology to help you identify terms, commands, and keyboard entries. The typographical conventions follow these guidelines:

Table 1-1 Typographical Conventions

Item	Example
MENU NAMES	SETTINGS
Menu Selections	Divide Line
Command Names	Gridset
<i>Dialog Box Titles</i>	<i>Layer Setup</i>
KEYBOARD KEYS	(CTRL, ESC, ENTER, TAB, SHIFT) KEY1 + KEY2: Press and hold KEY1 then press KEY2 KEY1, KEY2: Press and release KEY1, then press KEY2
Special function keys	[f1]

Mouse Conventions

Most mouse configurations (single-button or multiple button) that will work within the Windows environment will work with Terramodel. For a multiple-button mouse, the left button is always the primary button and the right button is considered the secondary button (unless you have configured it differently). The following terminology is used in conjunction with mouse movements:

Table 1-2 Mouse Conventions

Convention	Action
Click	Press and release the primary mouse button without moving the mouse.
Double-click	Press and release the primary mouse button two times quickly, without moving the mouse.
Point	Position the tip of the mouse pointer over the object you want to point to on the screen.
Drag	Point, then press and hold the primary mouse button while you move the mouse.

How to Use Terramodel Documentation

This section describes what documentation is available and how to best use the Terramodel documentation.

User Guide

The book you are now reading is the User Guide and is the proper place to begin. The *User Guide* provides an outline for the path of least resistance in learning Terramodel. Each chapter is a stepping stone of information, covering what you need to know as you progress from novice to experienced user to expert.

Read this book from start to finish.

Status bar

When running Terramodel, information about the current command is shown in the status bar. The status bar gives you a small hint about what each command will do. Use this resource when using Terramodel to determine if a command might be useful for a specific task.

On-Line Help System

Every command is documented in the On-Line help system. This documentation describes every option of the command but not how to use multiple commands to get a job done. While in a command press F1 or the HELP command button to get help about the command. Some commands have extensive documentation so continue to look at the next help page until you get to the next command. The help system is updated with every release of Terramodel and is your most accurate source of detailed information about each command. Use this resource when you need more information about a particular command as you use Terramodel.

Instant Expert On-line Guides

We have included several Instant Expert on-line guides as part of the Terramodel CD. These are only available in English. As you work through the tutorials in the guides, you'll learn the Terramodel approach to solving specific problems. Read the guides on-line from the Terramodel CD, or print them on your printer. These documents are in Adobe's Portable Document Format (PDF) (Version 4 is required.). If you do not have Adobe Acrobat Reader installed, you may install it from the setup program on the Terramodel CD.

The following files are the instant expert guides.

- [CD drive]:\DOCUMENTATION\EN\IEGST.PDF Getting Started Using Terramodel.
- [CD drive]:\DOCUMENTATION\EN\IECCM.PDF Creating a Contour Map.
- [CD drive]:\DOCUMENTATION\EN\IEGRA.PDF Grading Projects.
- [CD drive]:\DOCUMENTATION\EN\IERDD.PDF Roadway-Data Processing.

The *Terramodel User Guide* is also available in PDF format.

- [CD drive]:\DOCUMENTATION\EN\TMW_UG.PDF Terramodel User Guide.

Telephone support and Training

Trimble offers telephone support and training programs as described later in this chapter. These programs work together with the documentation to assure that you get the most productivity from Terramodel.

www.trimble.com

The Trimble website, www.trimble.com, contains additional resources that are updated from time to time. This site has answers to frequently asked questions and updated information about product support, training and all of the related Trimble software products. Clients under product support can download files and submit questions.

TML On-Line help

For advanced users Terramodel offers a complete programming language. Terramodel Macro Language (TML). Many of the commands you will be using are actually written in TML and can be customized by the end user. When you install Terramodel the TML help system will also be installed and can be used to learn about the power of TML.

Overview Of Terramodel

Terramodel provides the tools for surveyors, engineers, contractors and photogrammetrists to produce complete survey and design drawings. With it, you will be able to:

- Import survey data directly into a project file and reduce and balance field notes.
- Produce residential or commercial site plans.
- Contour sites.
- Perform site grading.
- Design storm drainage and sanitary sewer systems.
- Perform simple or complex roadway design.
- Draft plans.
- Perform hydrology studies and produce complete, submittal-ready reports.

This overview will introduce you to the modules and commands that comprise Terramodel.

Terramodel Modules

You can customize Terramodel by purchasing modules that accomplish survey and design functions that are specific to your areas of need. Each module gives you access to the commands related to that module. Modules are integrated in Terramodel, and require no additional installation. Table 1–3 lists each module and its functions.

Table 1-3 Terramodel Modules—Functions and Uses

Module	Function and Uses
Field Data	Provides basic functionality to manipulate field data.
CAD	Provides access to the basic CAD functionality of Terramodel. This module is required by all other modules except Field Data.
Contour	Creates contour lines from 3D data points.
COGO (Coordinate Geometry)	Includes advanced coordinate geometry commands used in surveying and site design.
Roadway Design	Provides tools for horizontal and vertical alignment design, and earthwork computations.
Site Design	Includes terrain modeling for original and design DTMs, projects one terrain model onto another, merges two terrain models into a composite terrain model, computes volumes between DTMs and performs simple roadway design.
Hydrology	Computes drainage areas, calculates pre-developed and developed storm hydrographs, provides pond design and volumes calculations, routes hydrographs through ponds and reaches.
Sewer Design & Analysis	Provides tools for analyzing, designing, and drafting storm drainage and sanitary sewer systems.
Image Manager	Provides the ability to view and plot image files as a background for the design data.
ASAP	Automated Sheet Assembly and Production (ASAP). Automates the creation and management of a set of plans.

Field Data Module Commands

The Field Data module allows you to use Terramodel to manipulate field data. You can open new and existing project files, import field data, compute preliminary coordinates, and plot the results or save the results in a project file. The commands associated with the Field Data module do not require a hardware lock. The commands listed in the following table are available in the Field Data module as well as all other modules:

Table 1–4 Field Data Commands

Module	Commands		
Field Data	All	Id	ProjInfo
	Area	IdAngle	RDE
	AutoDraft	Import	Recenter
	AutoSave	LabelPoint	Redo
	ClearMess	LayerSet	Redraw
	Close	Linetype	ReLayer
	Color	LinetypeSet	Rotate
	Command	Magnify	Save
	ConvSet	MapPoints	SaveAs
	Coords	MeasUnit	Scale
	Delete	MenuCfg	Search
	DisplaySet	Move	SnapSet
	Distance	Move	Staked
	Edit	Name	System
	Elevation	New	Undo
	Exit	ObjSnap	UnitsSet
	Export	Off	ViewRotate
	Favorites	On	ViewSet
	GDMDir	Open	VRecall
	GeomInq	Palette	Vsave
GeoSys	Pan	Zoom	
GFE	Previous		

Commands Secured to Modules

Some of the commands in Terramodel can be accessed only if you have purchased a particular module. When you run a command that is associated with a module, Terramodel checks that the hardware lock is currently authorized for that module. The following table lists Terramodel modules and the commands that are secured to them.

The Terramodel macro language (TML) is secured to the CAD module as are all commands written in TML.

The **TVLite** command is also secured to the CAD module. This allows anyone with a CAD module to create colored orthographic images of a 3D surface to display behind their Terramodel design.

Table 1-5 Secured Commands

Module	Commands
Cad	The Cad module is the basic secured level. All commands not specifically designated as a part of the Field Data module in the previous table or secured to other modules in this table are secured to the CAD module.
Contour	Import (DEM option) Profile AvgEnd Contour Xsection ContourAtElev XsOut Spot RemoveLinks DtmGrid ContourSet DtmArea Xsheet DtmEdge XsLabel DtmMatch DtmPts Slope Qprofile

Module	Commands		
Cogo (Coordinate Geometry)	TwoPL	Bbear	Coordcon
	ThreePC	Bfitline	CurveSol
	Bfitdurv	CdsProf	Levels
	TanCircle	CuldeSac	Level3w
	Ddist	Layout	Levellst
	Bdist	SegEdit	Resect
		Pstation	RTScale
		Parking	Sunstar
		PredArea	TriSol
	Traverse		
Site Design	DtmShot	Volume	Slice
	Design	CountorVol	SmpRoad
	Merge	3D	SmpTmple
Image Manager	Image (with Register, Clip, and Dynaimage functions)		
ASAP	Planset	Pssheet	Psindex
	Pstype	Psman	Objreg
	Pset		
Hydrology	Hdefs	Hgen	Pondsum
	Hcover	Hrlist	Pondvol
	DtmDrain	Hlist	Pondlist
	PtDrain	Hdraw	Rain
	Hcen	Reach	Raint
	Basin	Grout	Rgraph
	Hareab	Hroutr	
	Hblist	Pondout	

Module	Commands		
Sewer Design & Analysis	Profile	Pipesclq	Piperslt
	Pipedef	Pipev	Pipelbr
	Pipedsgn	Pipesolv	Pipemaxv
	Node	Pipell	Pipeminv
	Pipe	Pipechrt	Pipewsp
	Pipebycr	Pipedall	Pipesum
	Pipeq	Pipedraw	
	Pipecalq	Pipexing	
Roadway Design	AveEnd	Tmanter	PhaseMan
	Materials	Template	Phase
	PhaseName	Smanager	Masshaul
	ShapeClass	Subgrade	RoadSigPoint
	ShapeMan	Superelv	MassImport
	Shape	Skip	RoadDtm
	HALManager	RDXLines	LabelRoadHal
	VALManager	XsectionMan	LabelRoadVal
	SlopeManager	XsectionEdt	RoadRpt
	RDValEdit	XsectionDigi	Cstake
	RDValDesCrt	Roadway	Staking
	RDDesignSet	RDDesignCrit	EarthWrk
	RoadGrid	RoadSpot	RdsEcho
	RdsDtmSet	DesignElev	RDX
	RoadJob	Scalegrid	SuperPlot
	Surface	RoadMat	

On-Line Help

You can access information about Terramodel commands from the Terramodel Help program item in the Terramodel folder, or directly by choosing Help from the Terramodel menu. You may also press the F1 key to go directly to the help system.

Also included in the Terramodel folder is a TML help system, which provides a detailed explanation of how to write and modify TMLs.

Telephone Technical Support

Trimble Technical Support will respond to your software issues with knowledgeable support by engineers and surveyors who not only understand the software, but understand your business as well. Our fast response time will help you keep your projects on schedule. At Trimble, we believe the support we provide is just as important as any software feature.

Following your initial free support period, you can purchase annual support coverage. As part of our support program, you will receive free software updates during your coverage period as well as access to our unlimited toll-free telephone support and the support section of our web site. This will allow you to stay on top of the latest technology. Our Technical Support Staff is available 8:30AM to 5:30PM EST, Monday-Friday with later hours for our West Coast Clients.

You can use any of the following means to contact our Technical Support Staff:

Phone:	1-800-767-4822 or 1-408-481-6940
Address:	7403 Church Rand Blvd. Suite 100 Westminster Colorado 80021
E-Mail Address:	trimble_support@trimble.com
Web Site:	http://www.Trimble.com

Terramodel Training

Trimble's training programs are designed to cut learning curves and reduce wasted, non-billable hours. Our staff consists of civil engineers and surveyors who have used Terramodel in the field and understand real-world applications and the Terramodel tools you are using. You can learn in one day from our experts what might take you 10 to 12 days to learn on your own.

Our training programs are structured to meet the needs of individual clients. Our *Seminars* provide instruction on a pre-determined set of topics in a classroom setting. Sessions are held at various locations throughout the country, are from 1 to 3 days in length and very flexible. You will be notified by mail when a session will be held at a location near you.

You can also take advantage of our *Hands-On* training program. These classes provide even more thorough instruction on covered topics. Class size is limited and each student has their own computer.

Our training experts will also travel to you for customized, *On-Site* training. You decide the number of employees that will attend, the topics and pace. If you have a large group, this option is the most cost-effective. For information on our training programs, call 1-937-233-8921.

Minimum Requirements

Before you install Terramodel, make sure that your computer is equipped as follows:

Processor	Pentium, PC-compatible.
Operating System	Windows 95, 98, ME or Windows NT 4.0, 2000, XP or higher.
RAM	For Windows 95/98/ME: 32 megabytes. For Windows NT/2000: 64 megabytes, larger files require more memory.
Hard Drive	300 megabytes (minimum) of free space.
Graphics Card	800x600, 256 color minimum.
CD-ROM Drive	Required for installation of Terramodel.
Mouse	Microsoft-compatible.

Computer aided design and visualization are extremely demanding on the computer processor, memory, and your graphics adapter. For huge data sets, the fastest processor and graphics adapter available combined with the maximum amount of memory is recommended.

Installing Terramodel

The following instructions are written assuming you are familiar with basic Windows operations, such as using Explorer, moving between application windows, and using your mouse and keyboard.

Your Terramodel package includes the *Terramodel User Guide*, a hardware lock, and a CD containing the files necessary to install Terramodel. Close any open Windows applications before installing Terramodel.

Terramodel creates sub-folders within the installation folder and organizes files within these folders. The sub-folders are:

```
installation folder\shared\  
installation folder\shared\blocks  
installation folder\shared\fonts
```

installation folder\shared\fonts\Acad_Files
installation folder\shared\Formats
installation folder\shared\ImportExport
installation folder\shared\local
installation folder\shared\lscape
installation folder\shared\mentor.dir
installation folder\Terramodel\
installation folder\Terramodel\local
installation folder\Terramodel\tmls
installation folder\Samples\Autodraft
installation folder\Samples\Geodimeter
installation folder\Samples\lscape
installation folder\Samples\Trimble

To install Terramodel:

For Windows NT, 2000 and XP you must have administrator privilege for installation.

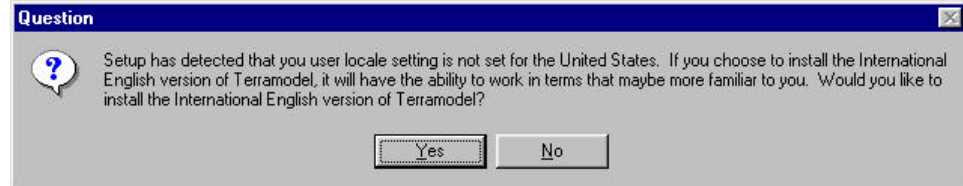
1. Insert the Terramodel CD in the CD-ROM drive.
2. The CD will start automatically; follow the installation instructions.

If AutoPlay is turned-off:

1. Click the Start button.
2. Click Run.
3. Type the CD-ROM drive letter followed by :setup in the Command Line box.
4. Click OK.

The Terramodel setup program guides you through the installation.

If you have your Windows Regional settings to any version of English other than “English United States” then the international English language version can be installed. After you have specified the installation folder you will be prompted:



The international version uses the term “Chainage” rather than “Station” for menu selections and reports. Command names remain the same regardless of the version installed. Metric based project files are installed as the default.

During a new installation you will be prompted for the location of a Custom configuration file. This is an advanced feature that enables the user to place their configuration files in a specific location. First time users should accept the default location (for English US settings this will be C:\Program Files\Trimble\Terramodel\Locale\English (US)). When any location is specified, other than the default, an environment variable is set which enables Terramodel to find the alternate location on startup. Manual configuration for this feature is described later in this chapter. An environment variable in Windows 95, 98 and ME is set by modifying the autoexec.bat file and requires that the user restart Windows.

When all files have been copied and the installation is complete, the installation program tells you that Terramodel was installed successfully. The Terramodel icon is placed in a new program folder called Trimble Office\Terramodel.

Hardware Lock Drivers

The setup program automatically installs the hardware lock drivers. For Windows NT, 2000, and XP you must have administrator privileges to install the driver. It is recommended that you install the drivers manually if you encounter any problems with your computer recognizing the hardware lock and/or printer.

To install the hardware lock drivers manually:

1. Insert the Terramodel CD into the CD-ROM drive.
2. Click the Start button, and click Run. The *Run* dialog box appears.
3. Type the CD-ROM drive letter followed by `:\\lockdriver\\setup` and click OK. Follow the onscreen instructions.
4. Some systems may require that you restart your computer.

Uninstalling Terramodel

To uninstall Terramodel, run the Terramodel setup again and select the uninstall option.

The Terramodel Search Path

Terramodel gives you control over how it searches for files through the folders you specify in what is known as the Terramodel search path or TSP. When Terramodel searches for the files it needs to load for your project, it first looks in the project folder. It then looks through a list of customization folders that you have specified. Finally, it looks in the installation folder.

Terramodel Folders

Project Folder

The project folder is the folder where the currently open project file is located. Place all supporting files specifically related to the project file in this folder. Supporting files typically include ASCII points files; as well as files imported from or exported to other formats. Other files you may wish to place in this folder include image files and related or reference project files.

The currently opened project file defines the project folder.

Customization Folders

A customization folder is a folder you create to keep Terramodel files that you have customized that are not specific to a single project. Place the files that apply to all projects and that you have customized to reflect your corporate standards in these folders.

Typical files may include prototype project files (*.PRO), block files (*.BLK), *.INI files, and *.MAP files. You can have as many customization folders as you find useful.

The TSP setting within the TMODWIN.INI file determines the customization folders as described below.

Installation Folder

The installation folder is the folder in which Terramodel is installed. It and its subfolders contain the executable files and all of the data files needed for the program's operation. It is important to note that the files contained within this folder and its subfolders are subject to replacement during the installation of a program upgrade. Therefore, it is unwise for the user to edit the files contained in this folder, as a means of customizing Terramodel to meet his needs, and to leave them there.

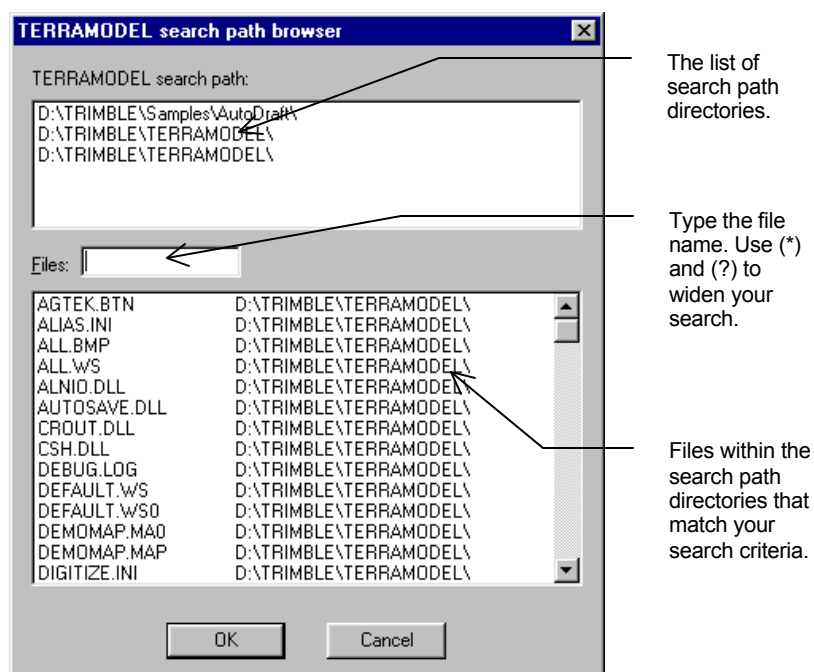
The location of the currently running Terramodel program determines the installation folder.

The TSP Command

The TSP (Terramodel Search Path) command lets you see your files the way Terramodel sees your files. When you run the TSP command, Terramodel lists them as it finds them. If you have a file in several different folders, Terramodel will list only the first one it finds.

To identify files accessible by Terramodel:

1. Type TSP into the command line. The Terramodel *Search Path Browser* dialog box appears.



2. To list a file of a particular name, type the file name into the Files box. Use wildcard characters for a more general listing of multiple files: type asterisk (*) for any number of characters or question mark (?) for a single character.

How to Use Customization Folders

Customization folders can be used to provide better control over how you customize Terramodel. This is an optional advanced feature.

If you followed the normal procedures when installing Terramodel, it is now located in a folder on your C drive named \Program Files\Trimble, i.e. the path to the Terramodel software and its related files is C:\Program Files\Trimble. Any changes that you make to the files located in this folder are subject to being lost the next time you install an updated version of the program.

The solution to this potential problem is simply to place copies of the files that you customize in a separate folder, and to instruct Terramodel to always use the copy you modified instead of the one contained in the C:\Program Files\Trimble folder.

Create the customization folder

From Windows Explorer, create a new folder on the C drive named \TMCUSTOM.

Now whenever you customize one of Terramodel's files you can place a copy of that file in the TMCUSTOM folder and modify that copy instead of the original. Another advantage of this procedure is that in the unlikely event your customization efforts don't go as planned; you will not have altered the original file. The next section tells you how to instruct Terramodel to use your modified copy instead of the one it would normally use.

You'll find numerous uses for the TMCUSTOM folder. In addition to containing just customized copies of files that were shipped with Terramodel, you can also use it to store external blocks that you create. If you wish to create a library of standard details and symbols, these can be created as blocks and stored in the TMCUSTOM folder.

It's always a good idea to keep the customization and data files that you create, separate from those that are shipped with Terramodel or any software product.

Your prototype project files, plan set configuration setting files, custom linetype and font files, custom TML files, etc., should all be stored there. If you find that you have a large number of one particular type of file located within the TMCUSTOM folder, you may wish to consider organizing these into subfolders. As an example, you might wish to create a folder named C:\TMCUSTOM\BLOCKS to store your external block files if you have a lot of these. It's totally up to you how you wish to organize this data. The only rule is that you must instruct Terramodel where to find things. That procedure is covered in the next section.

Edit TMODWIN.INI

When using customization folders the locations of these folders must be manually established in the TMODWIN.INI file. If you specified a custom configuration folder during installation you should use

1. Make sure Terramodel is not running.
2. If you have not previously created a customization folder, use the Windows Explorer to create the folder C:\TMCUSTOM.
3. Use the Windows Explorer to copy the files C:\Program Files\Trimble\Terramodel\Locale\English (US) and C:\Program Files\Trimble\Shared\Locale\English (US) from the Terramodel installation folders to the TMCUSTOM folder. Note that the name of the folder varies with the language version installed.
4. Open the C:\TMCUSTOM\TMODWIN.INI file in a word processing program such as Word, or WordPad that can edit ASCII text files.
5. Do a search for the word "System" and start a new line after the "...location of data files" line.
6. Type

```
TSP=C:\TMCUSTOM\;C:\TMCUSTOM\BLOCKS
```

into the new line where C:\TMCUSTOM\;C:\TMCUSTOM\BLOCKS represent customization folders separated by a ";". You may list as many customization folders as desired, but be sure to include the entire path.

7. Save the TMODWIN.INI file.

Set the TMODWIN Environment Variable

Now that you have placed the TMODWIN.INI file in a custom folder you must provide a way for Terramodel to find this file when the program starts. Setting what is known as an environment variable enables Terramodel to find the proper file.

For Windows 95/98/ME:

1. Open your c:\autoexec.bat file in the word processing program.
2. Type

```
SET TMODWIN=C:\TMCUSTOM\TMODWIN.INI
```

into a new line in the file where C:\TMCUSTOM\ represents the folder where you have placed your customized TMODWIN.INI file. This step tells Terramodel which TMODWIN.INI file to use if you have more than one.

3. Save the autoexec.bat file, close the word processing program, and restart your computer.

For Windows 2000:

1. Go to Settings\control panel\system and click the “advanced” tab
2. Add a new system variable:

You have now completed the changes required to effectively use the Terramodel search path.

The Terramodel Hardware Lock

Terramodel's program security (and access to the various modules) is provided by a hardware lock that is designed to attach to your computer's 25-pin female parallel port. Since the lock is invisible to (most) other software, it can remain in place while you run other programs. Only one lock is provided with each software package. If the lock is lost, stolen or destroyed, you must purchase another copy of the software to replace it. We strongly suggest that you insure the lock for the full replacement cost of the software.

Your software License Agreement allows you to use one copy of the software on only one computer at a time. To use the software on another computer, you must move the hardware lock.

To install the hardware lock:

Simply attach its male end to the parallel port on the back of your computer (if you have more than one parallel port, attach it to one that is not in use).

If you have a printer attached to the parallel port:

Attach the lock directly to the parallel port, then attach the printer to the other end of the lock.

If you have other hardware locks on the printer port:

Install the Terramodel hardware lock on the back of the other locks. (Sometimes the order of the locks on the parallel port affects their operation. If one arrangement does not work, change the placement of the locks until all software programs are operational.)



You must not remove the hardware lock while Terramodel is running! If you do you will be required to replace the lock to continue any command other than end or save.

Terramodel Opening State

Terramodel Without a Hardware Lock

When you run Terramodel without a hardware lock, you have a limited selection of Terramodel commands (*see the list of Field Data module commands in the Introduction*). In the absence of a hardware lock, Terramodel allows only a plan view and this plan view cannot be closed, either by command or by closing the view window. The **Vnew** command is not available when Terramodel is run without a hardware lock, and new views cannot be created.

All imported point objects and traverse sets created by the Raw Data Editor will appear in the plan view.

Note: Some settings in Units Settings, Display Settings and View Settings are not grayed-out when Terramodel is run without a hardware lock, even though they are not relevant to any available command.

Terramodel with a Hardware Lock

When you open an existing project in Terramodel with a hardware lock, a plan, profile, and sheet view are created and the plan view is maximized. To access the profile or sheet view, minimize the plan view. The minimized profile or sheet views appear in the bottom left corner.

When you create a new project in Terramodel, only a plan view is created. This plan view is maximized. You must create any other views that you need with the **Vnew** command.

The first time you start Terramodel, the coordinate window appears towards the bottom right side of your screen. This location varies with your screen's pixel display capacity. If you don't like the position, change it. The coordinate window remembers its last position. Click on its title bar and drag it to a new location. When complete click in the command line.

Configure These Critical Settings

Terramodel has many settings that enable you to customize its operation for local standards and terminology.

Caution!

Critical information such as bearings and distances can be entered and reported in multiple formats. Some settings have formats that are similar or identical and hence difficult or impossible to interpret without knowing the project settings. For instance is the bearing: 35 . 25 50 , 35°25' 50" or is it 35.2550° ?

**It is the user responsibility
to understand and select the desired settings.**

Before you start working establish the settings for the commands shown below. Run each of the commands and review the help system for specific instructions. Please note that not all of the program settings are listed.

Settings are saved either in a project file or external to a project file. External settings are intended to remain the same regardless of the project file in use. Typically these may relate to user preference and the computer system in use. Settings saved in a project file are intended to remain the same in the event the project is moved from system to system.

System Settings

These settings are saved in the tmodwin.ini file and remain the same regardless of the project file.

File\System configuration\Autosave (**AutoSave**)

File\System configuration\Command alias (**Alias**)

Settings\Convention settings (**ConvSet**)

Window\Coordinate scroll (**Coords**)

File\System configuration\Favorites (**Favorites**)

File\System configuration\Palett + colormap (**Palette**)

File\System configuration\System (**System**)

Keyboard only or select from the Toolbar (**ToolBox**)

\File\Plot setup\Plot params (**PlotSet**)

\File\Plot setup\Plotter setup (**PlotterSet**)

\File\Plot setup\Carousel setup (**Carousel**)

Settings in the File\System configuration\System (**System**) command are saved in both the ToolPak (p3server.ini) and Client (tmodwin.ini) ini files as shown in the command's dialog box. The prototype project setting of this command is established during the installation of Terramodel based on the selected localization option.

In the Settings\Convention settings (**ConvSet**) command the rise/run option is saved in the project file, all other settings are saved in the tmodwin.ini file.

The following commands save their settings in the current project file. A prototype project file is used as the basis when a new file is created. The prototype file is specified in the File\System configuration\System (**System**) command. To establish project settings for all new project files simply open the prototype project file, establish the desired settings and save the project.

Project Settings

These settings are saved in each project file.

- \Settings\Abbreviations (**AbbrevSet**)
- \Settings\Contour settings (**ContourSet**)
- Settings\Convention settings (**ConvSet**)
- \Settings\Display settings (**DisplaySet**)
- \Settings\Draft settings (**DraftSet**)
- \Settings\Grid settings (**GridSet**)
- \Settings\GPS Geodetic System (**GeoSys**)
- \Draft\Label sets (**LabelSet**)
- \Draft\Label stations (**LabelSta**)
- \Draft\Label table (**LabelTable**)
- \Draft\Label points with text (**LabPt**)

- \Settings\Layer settings (**LayerSet**)
- \Settings\Linetype settings (**LinetypeSet**)
- \Settings\Link settings (**LinkSet**)
- \Settings\Running snaps (**ObjSnap**)
- \Settings\point settings (**PointSet**)
- \Settings\Snap settings (**SnapSet**)
- \Settings\Style settings (**StyleSet**)
- \View\View settings (**ViewSet**)
- File\System configuration\System (**System**)

A few commands maintain three sets of settings that change with the project units in use. When you change the measurement units in the \File\Measurement units (**MeasUnit**) command from feet to meters these settings change *without warning*.

Project Settings that Reflect Unit Settings

These settings are saved in each project file and change with the current measurement units (meters, feet or other).

- Settings\Design settings (**DesignSet**)
- Draft\Label grid (**LabelGrid**)
- Draft\Label HAL (**LabelHAL**)
- Draft\Label points with text (**LabPt**)
- \File\Measurement units (**MeasUnit**)
- Roads\Alignments\VAL design criteria (**RdValDesCrit**)
- Settings\Units settings (**UnitsSet**)

The Settings\Units settings (**UnitsSet**) command controls the basic format of distances, bearings, angles and other critical settings.

Configure these Settings for Localization

Out of the numerous Terramodel settings there are a few that typically vary from location to location. These settings are highlighted below.

Users should note that Terramodel has the flexibility to circumvent most settings with TML commands or by creating specific report formats. If in doubt, be sure to check your numbers!

X Y or Y X

Terramodel has a variety of commands where coordinates are entered or reported. The order of coordinates is controlled by the \View\View settings (**ViewSet**) command. Each view (plan, profile, sheet, etc.,) has its own settings.

Station and Chainage

In the early 1600's the Gunter's chain was invented. Its length is one-tenth of a furlong and is composed of 100 links. Measurements are recorded as so many chains + links. The Gunter's chain is unique in that the area in acres of any rectangular tract of land measured in chains is one-tenth of the product of the length and width.

In the US this system of recording distances is commonly referred to as stationing. In most of the rest of the world, this system is commonly referred to as chainage.

In Terramodel versions prior to 10.0 this change was handled by the **ConvSet** command. In Terramodel version 10.0 and later an alternate English language versions must be installed. This alternate version can only be installed when the Windows Regional Setting is other than English (United States).

This change in terminology is handled as with any other language translation. Menu selections and reports will reflect the change, however the actual command names remain the same for all languages.

Rise/Run or Run/Rise

The format for the entry and reporting of this slope definition is controlled by the Settings\Convention settings (**ConvSet**) command.

North, South, East Azimuths or Bearings

The zero direction for azimuth angles is set in the Settings\Units settings (**UnitsSet**) command. This command also sets the angle format to: degrees minutes seconds, decimal degrees, grads or mils.

Extended ASCII Characters

In the p3server.ini file, the setting [Show8BitChars] is set to either 0 or 1. When set to one editing text objects with ASCII characters over 127 will show the font character. When set to zero the ASCII equivalent will be shown with the "\", ie. \176.

Setup

During the installation of Terramodel you have the ability to select one of several regions. General settings are implemented for the selected region. These selections can be revised without having to re-install the program. You should review the commands listed in the *Configure These Critical Settings* section to assure you have selected the desired settings.

- **ConvSet** –Terminology (Station, Chainage, etc...)
- **System** –Prototype (tmodel.pro, tm_eu.pro, tm_au.pro)

Each prototype project file has been edited to establish the most common settings for each region. You will want to check the settings and add your own changes for your office standards.

In the tmodwin.ini file the default workspace for the toolbox is specified. In the section [IconTOOL] the file is set to default.ws, tm_eu.ws or tm_au.ws. This setting is modified by running the ToolBox and opening a Workspace.

In the p3server.ini file, the setting [Show8BitChars] is set to either 0 or 1. When set to one editing text objects with ASCII characters over 127 will show the font character. When set to zero the ASCII equivalent will be shown with the “\”, ie. \176.

System Configuration

To set the configuration within the **System** command:

1. Open the **FILE** menu.
2. Click **Config System**, then click **System**. The *System Configuration* dialog box opens.

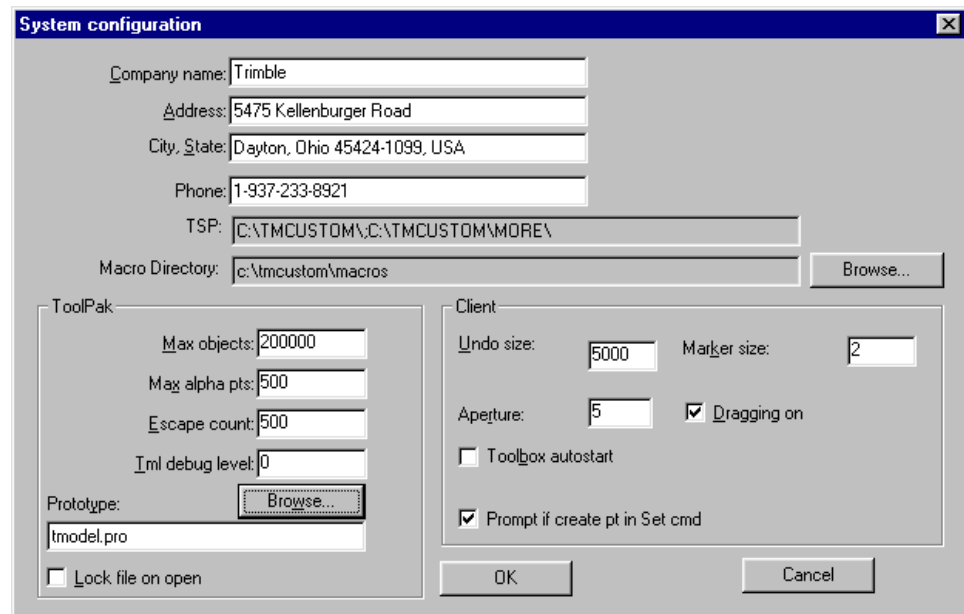


Figure 1–1 System Configuration Dialog Box

Company Information

This section of the dialog box lets you enter the information that will appear in report or listing headings, and on printouts.

TSP (Terramodel Search Path)

The Terramodel Search Path determines the order Terramodel searches your folders. By default, the TSP starts with your current project folder and ends with the Terramodel installation folder. The TSP is set by directly editing the TMODWIN.INI file as described previously in this chapter.

Macro Folder

The Macro Folder determines where Terramodel will save and search for keyboard macro files. Recorded macros must be placed in this folder in order to be played.

Server

This section of the dialog box configures the settings that are stored in the P3SERVER.INI file, which is activated each time Terramodel is initialized.

- **Max objects**—This parameter lets you enter an integer to set the maximum number of objects (points, sets, polylines, text, etc.) that the current project file can contain. Commands that create new objects will not be completed if this maximum is exceeded during the command execution.

The number of objects you can have in a project file is directly related to the amount of random access memory (RAM) and disk swap space in your computer. A good rule of thumb to use is 100,000 objects for the first 32 megabytes of RAM, then 6000 more objects for each megabyte of RAM after the first thirty two. Although Terramodel automatically uses disk swap space, due the slower speed of the disk as compared to RAM, it is not recommended that swap size be considered. The amount of memory required by an object can vary greatly due to its name, number of points, number of vertices, etc. and hence there is no hard and fast rule to follow. The following table is a good rule of thumb for most users.

Table 1-6 Recommended maximum number of objects

RAM	# of objects
32 MEG	100,000
64 MEG	300,000
128 MEG	700,000
512 MEG	3,000,000

The maximum number for this setting is 8,000,000. For each object Terramodel pre-allocates about 20 bytes for initialization. Setting the maximum number of objects too high wastes RAM that can be used for object storage. Note that Windows currently can not use more than 1024 MEG for Terramodel data.

This setting establishes the value for the **maxRec** parameter in the [DBTYPE] section of the P3SERVER.INI file.

- **Max Alpha Pts**—This integer control lets you enter the maximum amount of alphanumeric point ids that your projects can contain. Increasing the size of this setting increases the memory requirements. If you do not plan to use alphanumeric point ids, set this value to zero. This value is set to 0 by default.
- **Escape count**—This parameter lets you enter an integer to establish the number of objects that Terramodel will draw during a screen regeneration before it checks to see if you have pressed the ESC key. This option lets you stop a screen redraw if you have a large number of objects that could potentially require a lot of time to display. Suggested values are from 50-200 for 486-based computers, and 100-1000 for Pentium-based computers.

This setting establishes the value for the **escCount** parameter in the [TMTYPE] section of the P3SERVER.INI file, and takes effect each time you start Terramodel.

- **Tml debug level**—This **setting** establishes the value for the **TmlDebug** parameter in the [User] section of the P3SERVER.INI file, and takes effect each time you start Terramodel. The use of this setting is described in the TML help system and should only be changed when writing TML commands.
- **Prototype**—This edit text box lets you enter the name of an existing file that will be used as a prototype for new files. When a new file is created a copy of the prototype file is used as the basis of the new file. In the event there are no project files available you can enter 0 (zero) as the file name and a new file will be created from the internal Terramodel defaults. For additional information on setting up

prototype files, see “Creating A Prototype Project File” in chapter 3, **How To Create, Save, and Retrieve Projects**, of the Terramodel *User Guide*.

- **Lock file on open**—This check box controls the creation of a locked status file. When enabled a **file** with the extension of .PLK will be created indicating that the file being opened is in use and can not be opened by another user. The lock file is an ASCII file with the version of the lock file, its status and the time the file was created. The lock file is remove when Terramodel properly closes the project file.



In the event the file is not properly closed, i.e. there is a loss of power, the .PLK file must be manually deleted.

Client

This section of the dialog box configures the settings that are stored in the TMODWIN.INI file, which is activated each time Terramodel is initialized.

- **Undo size**—This parameter lets you enter an integer to set the size (in number of objects) of the undo buffer. As objects are entered, deleted, or modified in the Terramodel database, they are also saved to this buffer so that action taken on them **can** be reversed. Since the amount of memory used by the buffer reduces the amount of memory available for the project file, we recommend setting this parameter to 5000 objects.

This setting establishes the value for the **UndoSize** parameter in the [System] section of the TMODWIN.INI file, and takes effect each time you start Terramodel. It can be changed at anytime from within a Terramodel project file, and is immediately effective.

- **Marker size**—This parameter lets you enter an integer to set the size in pixels of the ghosted diamond marker that Terramodel draws to indicate where a line has been **broken** with the **Break** command, etc.

This setting establishes the value for the **MarkerSize** parameter in the [System] section of the TMODWIN.INI file, and takes effect each time you start Terramodel. It can be changed at anytime from within a Terramodel project file, and is immediately effective.

- **Aperture**—This integer control is used to set the size of the aperture pick box in pixels. You may wish to vary this depending on the resolution of you graphics display. Common values are two to six.
- **Dragging on/off**—This check box lets you turn on and off the ability to graphically move and rotate objects. When checked, you can see your objects move and rotate dynamically as you move your cursor. When not checked, the objects will not appear to move on the screen until you complete the move or rotate command.

This setting establishes the value for the **DragMode** parameter in the [System] section of the TMODWIN.INI file, and takes effect each time you start Terramodel. It can be changed at anytime from within a Terramodel project file.

- **Toolbox auto start**—This check box lets Terramodel’s TOOLBOX (which allows you to **create** your own command menus) start and display toolboxes automatically each time you open a project file. When not checked, you must manually start TOOLBOX.
- **Prompt if create pt in Set cmd**—This check box lets you turn on and off a warning message that can occur if you are creating a point with the set command. When using the set command, if the free running snap mode is active it is possible to create points “on the fly”. Enabling this feature will warn you when this is happening.

This setting establishes the value for the **PromptSet** parameter in the [System] section of the TMODWIN.INI file.

The Terramodel Initialization Files

The TMODWIN.INI contains all of the configuration parameters that Terramodel needs to initialize itself within the Windows environment. Most of these sections are automatically updated as you make configuration changes within Terramodel, and do not need to be changed manually (exceptions include the sections on palettes, color mapping, and pen mapping). The parameters contained in this file remain the same for all loaded project files.

Chapter Two

The Terramodel Display



In this chapter you will learn the basics of the Microsoft Windows operating system as it applies to Terramodel. Terramodel closely adheres to the Windows conventions and extends this system to include controls designed to meet the needs of a graphical CAD application. For those of you who are already familiar with Windows, portions of this chapter will seem more like a review. However, this chapter describes the use of important extensions that function throughout Terramodel, and should prove useful even for the experienced Windows user.



Windows Basics

Views (Windows)

Views in Terramodel allow you to look at multiple areas or different perspectives of your project simultaneously. You can copy or move objects between views and even move the views themselves around on the screen. Table 2-1, below, lists the various methods used to manipulate views, by mouse or keyboard.

Table 2-1 View Controls

To:		
Select a view.	Click the desired view.	Use CTRL+TAB to move to the desired view.
Open a view.	Double-click the desired view icon.	Press ALT, ENTER.
Maximize the view after you have opened it.	Click the view Control menu, then click the Maximize option, or click the Maximize button in the upper right corner of the view window.	Press ALT, HYPHEN then X.
Move a view to a different location on the graphics screen.	Click the view title bar with your mouse and drag the window to a new location, then release the mouse button.	Press ALT, HYPHEN then M. Use the arrow keys to move the view to the desired location; press ENTER to accept.
Size a view.	Click any of the four sides or any corner of the view and drag until the desired size is reached, then release.	Press ALT, HYPHEN then S. Use the arrow keys to move the cursor to a view edge. When the cursor changes, use the arrow keys to move the edge in the desired direction; press ENTER to accept.

To:		
Minimize a view (reduce it to an icon).	Click the Minimize button in the upper right corner of the window (if the view window is maximized, click the Restore button first).	Press ALT, HYPHEN then N.
Close a view.	Click the view Control menu, then click Close , or select VIEW/Close View from the menu bar.	Press CTRL + [F4]

Control Menus

Control menus appear in the upper left corners of the Terramodel screen and all dialog boxes and views. The contents of the control menus vary depending on the type of dialog box or window, but in general they contain commands which manipulate the windows themselves or end the command in use. Open any control menu by clicking the Control menu box once with your mouse, or following the shortcut keys below.

Control menu for Terramodel screens and views, type ALT+SPACEBAR to open. Table 2-2 Control Menu Options, describes the options available:

Table 2-2 Control Menu Options

Option	Function
Restore	Restores the window to former size after maximizing or minimizing
Move	Moves the window using keyboard arrow keys.
Size	Changes the size of the window using keyboard arrow keys.
Minimize	Reduces the window to an icon
Maximize	Enlarges the window to its maximum size.
Close	Closes the window or dialog box.
Next	Switches between views and view icons.

Dialog Boxes

A *dialog box* gives you a way to interact with the application. The dialog box usually contains one or more controls that display static text or graphics, or let you specify information.

In general, dialog box controls fall into these categories:

- Buttons.
- Edit text.
- Scroll bars.
- Static controls.
- List boxes.
- Combo boxes.
- Custom controls.

The sample dialog box below illustrates some of the various tools:

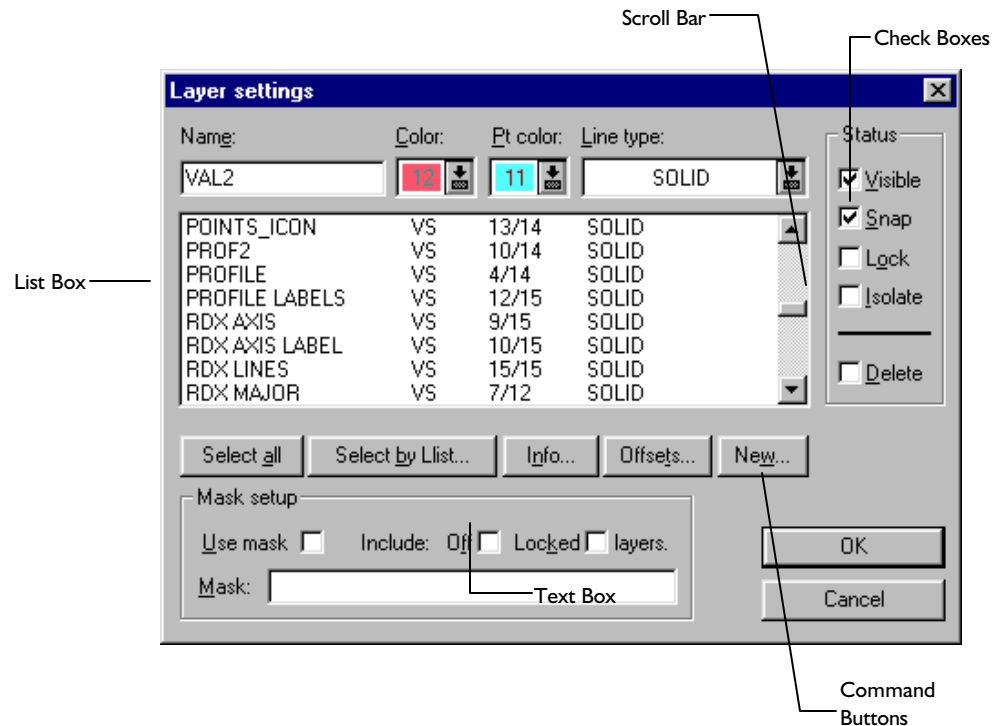


Figure 2-1 Dialog Box Tools

The dialog box tools are described below:

Table 2-3 Dialog Box Tools

Tool	Description
Command buttons	Execute a command or initiate an action. Example: OK, Cancel.
Text boxes	Require that you type a text response. Example: File Name, Layer Name.
Check boxes	Select or clear an option using a toggle to replace or remove an X from the box.
List boxes	Contain a list of choices from which you select one or more. Where there are more choices than lines in a box, a scroll bar allows you to scroll through your choices.

Basic Windows Controls

The basic controls available in Windows are also available in Terramodel . These controls include:

- Horizontal scroll bars.
- List box.
- Command button (Push button).
- Radio button.
- Vertical scroll bar.
- Combo box.
- Custom control.
- Check box.
- Group box.
- Edit text control.

If you aren't familiar with the use of these tools, a detailed discussion is included in your *Windows User's Guide*.

Key Focus

An important control concept in a Windows application is that of *key focus*. A Windows application has the ability to direct a key press to any one of many areas within the screen. In order to know the effects of most key presses, you must consider the current location of the focus. The focus is typically indicated by a flashing cursor (in the command line), a highlighted area (in the Toolbar or Menu bar) or a bold border on a command button.

Key Combinations

You can use many combinations of key presses to edit text, move the cursor, or open and close windows. The tables below list some Windows key combinations and how they interact with Terramodel .

Table 2-4 SYSTEM Key Combinations

System Key Combination	Function
ALT + [F4]	Quits Terramodel .
CTRL + [F4]	Closes the current view window.

Table 2-5 EDITING Key Combinations

Editing Key Combination	Function
SHIFT + INS	Pastes text from the Clipboard to the place indicated by the insertion point.
CTRL + INS	Copies selected text to the Clipboard.
ALT + BACKSPACE	Undo (undoes the last edit in the project file).






Table 2-6 DIALOG BOX Key Combinations

Dialog Box Key or Key Combination	Function
SPACEBAR	Selects an option button or cancels a selection. Places or removes a check in a check box. Opens a list in a drop-down list box.
CTRL + SLASH (/)	Selects all the items in a list box.
TAB	Moves the focus from option to option.
HOME	Moves to the first item in a list box or to the first character in a text box.
END	Moves to the last item in a list box or the last character in a text box.

Mouse Pointer Shapes

When you use your mouse, your cursor assumes various shapes to indicate what type of data entry Terramodel expects. Table 2-7 below illustrates the different mouse shapes that you will see:

Table 2-7 Mouse Pointer Shapes

Mouse Pointer	Indicates:
	The current view is invalid for selecting or entering data.
	Select box for selecting an object.
	The position of the cursor, reflected in the Coordinate Scroll.
	Computations or re-display in progress.
	Pointer to select commands from menus, move views and dialog boxes around the screen, minimize or maximize views.

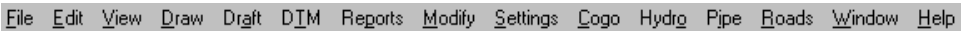
Common Button Controls

You will see a few common button controls on almost all of your dialog boxes and command bars. The functions of these buttons remain consistent throughout Terramodel :

OK or command name button	Completes the execution of a command after you've selected various options.
Cancel button	Escapes from a command without executing it, returning all options to their original status before the command was started.
Close button	Closes the command bar or dialog box progress. Due to possible changes during the command you are not assured that the original status can be restored.

The Menu Bar

You can access almost every command available in Terramodel from the Menu Bar at the top of the screen. Each menu, when selected with your mouse, expands to reveal a series of menu options. The command names, which correspond to the menu options, are displayed in the left section of the Status Bar when you select or place the focus on a menu option.



File Edit View Draw Draft DIM Reports Modify Settings Cogo Hydro Pipe Roads Window Help

Figure 2-2 The Terramodel Menu Bar



For a complete description of the menu options, see the Terramodel On-Line Help. Several commands are not located on any menu and must be entered from the keyboard. A complete listing of these command can be found in the On-Line Help system.

Change the focus from one menu option to another by holding down the SHIFT key and using the up and down arrow keys, or by holding down the left mouse button while moving the mouse through the menu options. When the focus is on a menu option, the center portion of the Status Bar displays a short description of that option.

The menu options follow a special convention to help you recognize the status of each option. A summary of menu conventions are listed in Table 2-8:

Table 2-8 Menu Conventions

Menu conventions	Description
Dimmed option	Not accessible at that time. In many cases you must select another option before this option can be selected.
Option followed by an ellipsis (...)	Opens a dialog box in which other options must be chosen before the command is completed.
Option preceded by a check mark (✓)	Already in operation. It can be toggled in and out of operation by selecting it to remove or replace the check.
Key combination	Option can be called using an abbreviated form, or “short cut”, which is shown to the right of the command (for example, ^Z is the abbreviation for Z oom). Most options can also be called by typing the underlined character within the option name, while the menu is visible.
Option followed by a triangle	Displays a cascading menu containing additional options.
Options with one or more letters underlined	Option can be started by typing just the underlined letter or letters.

Menu Configuration Options

The **MenuCfg** command allows you to select a different menu configuration file with a Windows file browser. After selecting a menu configuration file, the menus and menu options change according to the new file settings.

Terramodel is shipped with several menu configuration files, one for each product package. The first time Terramodel runs, it checks the current hardware lock setting to select the appropriate menu configuration file and updates the .INI file. If you purchase new Terramodel modules, you must use the **MenuCfg** command to change the menu configuration file accordingly.

Configuration File	Associated Module(s)
Spfielddata.m	Field Data module. This module does not require a hardware lock.
Spcad.m	Field Data, CAD module.
Spsurvey.m	Field Data, CAD, COGO and Contour modules.
Spdesign.m	Field Data, CAD, COGO, Contour and Site Design modules.
Spconstr.m	Field Data, CAD, COGO, Contour, Site Design, Roadway and ASAP modules.
Tmodwin.m	All Terramodel commands are listed in this configuration file.

You can create your own customized configuration files to eliminate commands that you don't want as menu options or add commands that you do. You can even have your own *User* menu. However, Terramodel still checks the hardware lock before running a command.

The Status Bar

The Status Bar displays information about a command. The information is displayed in three sections:

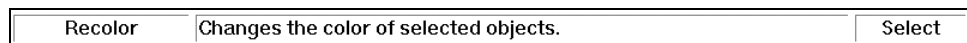


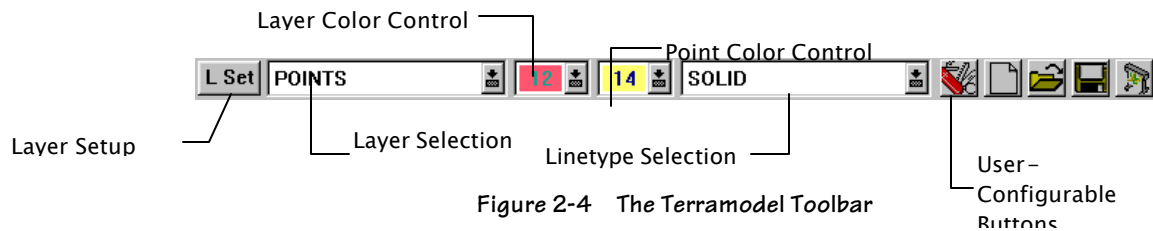
Figure 2-3 The Terramodel Status Bar

Status Bar Sections

Left Section	Displays the current command or the command corresponding to the menu option that has the key focus.
Center Section	Gives a one-line description of the command.
Right Section	Displays the Terramodel custom control in operation in the command. For information on custom controls, see chapter 6, <i>Using Terramodel Custom Controls</i> .

The Toolbar

Most commands in Terramodel place the objects they create on the current layer using the colors and linestyle of that layer. The Toolbar contains the tools to set the current layer name and its colors and linestyle. The Toolbar also allows you to assign commands to user-configurable buttons for easy access.

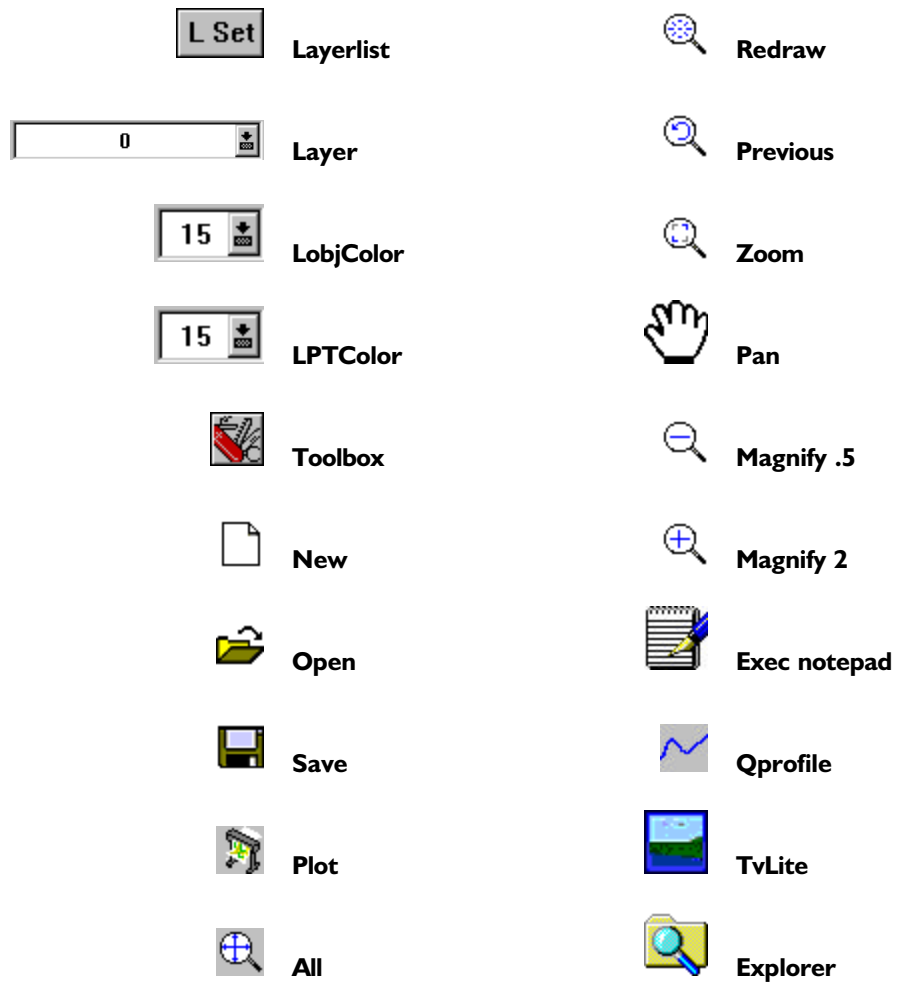


Toolbar Controls

L Set	Opens the <i>Layer Setup</i> dialog box. You may also use the LayerSet command.
Layer Selection	Lets you control which is the current layer. You may also use the Layer command.
Layer Color	Sets the color that will be assigned to any new objects (except for points) created on the current layer shown in the Layer Selection box. You may also use the LObjColor command.
Point Color	Sets the color of any new points created on the current layer shown in the Layer Selection box. You may also use the LPTColor command.
Linetype Selection	Changes the linetype of the current layer. You may also use the LLType command.
User-Configurable Buttons	You can configure the buttons at the right end of the Toolbar to call commands, as well as to display custom pictures, or <i>bitmaps</i> . See chapter 18, <i>Customizing Terramodel</i> , “Customizing the Terramodel Toolbar”, for details.

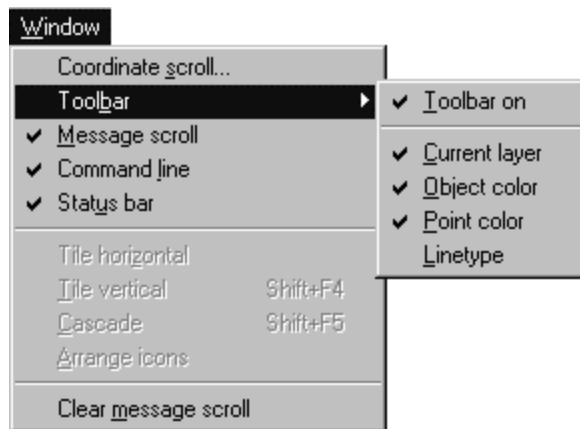
The Default Toolbar

The Terramodel toolbar, as shipped, has the following controls or icons:



Toolbar Toggles

Terramodel allows you to toggle the display of the following controls on the toolbar: *Current layer*, *Object color*, *Point color*, and *Linetype*. Select the corresponding *Window /Toolbar* option to toggle the checkmark on and off. A check by the item means the item is displayed. The absence of a check means the item is not displayed. The following Toolbar submenu configuration is the shipped default:



Layer Setup Dialog Box

The **L Set** button on the Toolbar starts the **LayerSet** command and opens the *Layer Setup* dialog box for assigning layer names, colors and linetypes to selected layers. You can also create or delete layers and set the visibility, and snap status for one or more layers.

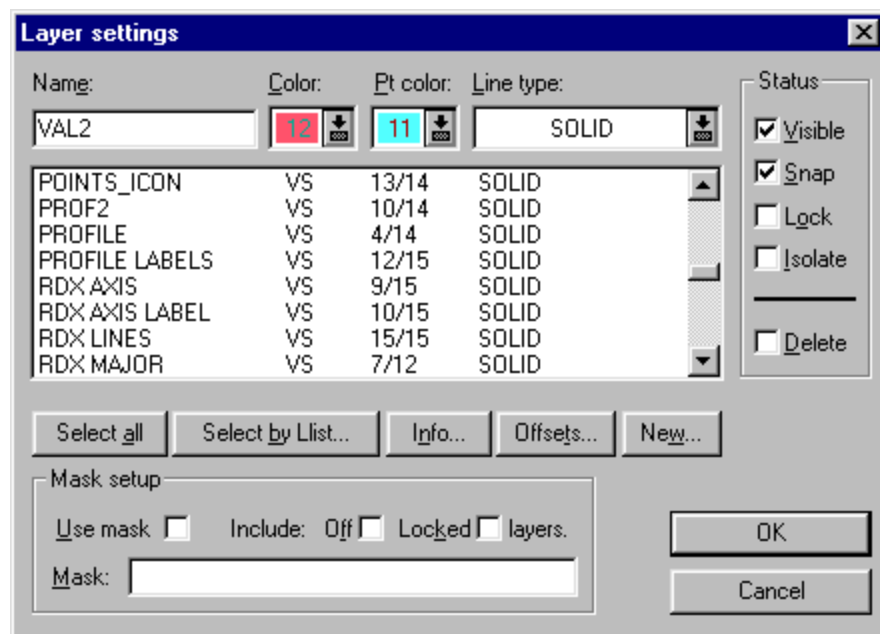


Figure 2-5 Layer Setup Dialog Box

Chapter 5, **Understanding Terramodel CAD**, “Working with Layers”, contains a complete discussion on how to use the features of the *Layer Setup* dialog box.

Layer Selection Control

Lets you control which layer is the current layer. When the focus is on this control in the Terramodel Toolbar you may select any object in the graphics window and its layer will be used as the current layer. Clicking the down arrow on the Toolbar opens the *Layer Selection* dialog box, which lists the layers in your project file.

Layer Selection Dialog Box

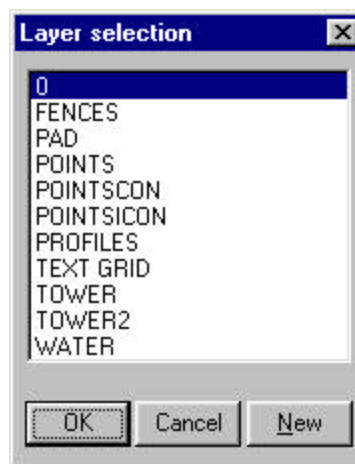


Figure 2-6 Layer Selection Dialog Box

The **Layer** command also opens the *Layer Selection* dialog box. Select from the list to set a current layer or click New to create a new layer.

Layer and Point Color Controls

When the focus is on either of the color numbers displayed in the Toolbar, you may select any object in the graphics window and its color will be used as the new layer or point color. Clicking the down arrow opens the *Color Selection* dialog box from which you can select any available color. Chapter 5, **Understanding Terramodel CAD**, “Working with Colors”, discusses in detail how to use the color controls and the *Color Selection* dialog box. The **LObjColor** command also opens the *Color Selection* dialog box and sets the layer color for the current layer. The **LPTColor** command also opens the *Color Selection* dialog box and sets the point color for the current layer.

Linetype Selection Control

Changes the linetype of the current layer. When the focus is on this control in the Terramodel Toolbar you may select any object in the graphics window and its linetype will be used as the current linetype. Clicking on the down arrow opens the *Linetype* dialog box.

Linetype Dialog Box

The *Linetype* dialog box lists the linetypes for the project file and lets you select a linetype.

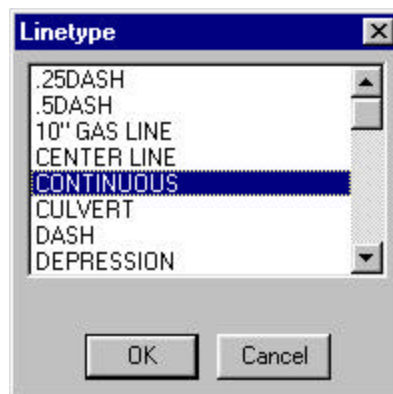


Figure 2-7 Linetype Dialog Box

For more information on linetype files and creating and selecting linetypes, see chapter 5, *Understanding Terramodel CAD*, “Working with Linetypes” and “Creating Custom Linetypes”.

The Coordinate Scroll

The Coordinate Scroll displays a variety of coordinate readout combinations to track the location of the cursor. Enable each option as desired and the appropriate values will be displayed when applicable. Note that the information displayed may vary depending on the current command, the current control type (locate, bearing, distance, offset, station, point, etc.) and the active view mode (plan, profile, sheet, cross section, etc.).

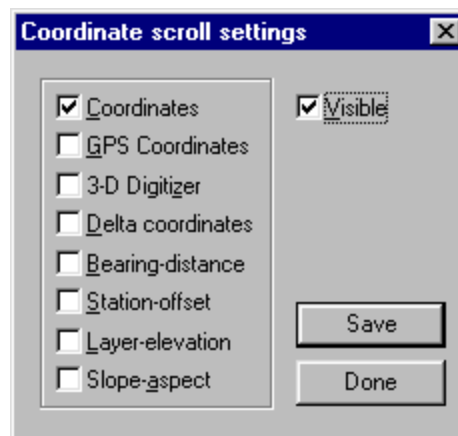


Figure 2-8 Coordinate Scroll

Coordinate Scroll Options

- | | |
|------------------------|---|
| Coordinates | When the cursor is in the plan view mode it displays its Northing and Easting. In the profile view it displays the station and elevation. In the cross section view it displays offset and elevation, and in the sheet view, X and Y. |
| Gps Coordinates | This option is reserved for future use. When properly connected to a real time GPS system this option will display the X,Y,Z coordinates of the GPS unit and the Cut/Fill distance to the current DTM layer. |
| 3-D Digitizer | This option is only used when the digitizer type responds with X, Y and Z values. |

Delta Coordinates	This option only responds when there is a “rubber band” cursor and shows the change in coordinate values from the anchor point to the cursor in the current view mode units.
Bearing–distance	This option only responds when there is a “rubber band” cursor. It shows the bearing and distance from the anchor point when in the plan view mode. In the profile or cross section views it shows the grade and distance.
Station–offset	This option only responds when there is a “rubber band” cursor and when a station control is active. The station and offset relative to an alignment is shown in the plan view mode. When the cursor is rubber banding in the plan view a single line showing the station will be shown in profile view and the offset in the cross section view. When rubber banding in the profile view mode the station and the difference in elevation from the selected alignment to the cursor is shown.
Layer–elevation	This option shows the current DTM layer and the elevation on the DTM at the cursor location. The current DTM layer can be reset by using the Spot command on the desired layer.
Slope–aspect	This option shows the slope of the TIN on the current DTM layer at the cursor location and the direction of the surface slope. This option must be used in conjunction with the above layer-elevation option, which establishes the current DTM layer.
Visible	The option controls the visibility of the coordinate scroll.
Save	This option saves the current settings to the TMODWIN.ini file and exits the command.
Done	This option keeps the current settings for the current session without updating the TMODWIN.ini file and exits the command.

Tips on using the **Coordinate Scroll**:

The default number of decimals used in the Coordinate Scroll is set to two. The accuracy of the cursor position is controlled by the screen or digitizer and hence two decimals is typically more than is needed. In the event you desire to vary this number you may add the following setting to the TMODWIN.INI file in the [CoordScroll] section of the file: `CoordNdec=x`, where x is the desired number of decimals.

Move the scroll by clicking anywhere inside its title bar (where the word “Coordinates” appears) and dragging the scroll to the desired location.

Reduce the scroll to an icon by clicking its minimize button.

Maximize the scroll by double-clicking the word *Coordinates* which appears with the icon.

The Command Bar

Much of the communication between you and Terramodel is relayed within the Command Bar area. When the focus is in the Command Bar, you can start a command by typing its full name (or its alias). The prompts that you are required to answer before the command is completed appear in this part of the screen, often in the form of list boxes that, once a selection is made, open dialog boxes with additional choices.

The Message Scroll

The Message Scroll displays information about your projects in response to commands that you have executed. If you have inquired about an object, you can retrieve its type, coordinates, color and layer from this area. It also shows screen regeneration times and the amount of time it takes to complete certain commands such as **Contour** and **Volume**.




Figure 2-9 The Terramodel Message Scroll

The scroll bar to the right allows you to scroll back to previous messages from earlier commands. You can scroll back for a maximum of 64K of messages which, for long sessions, may not return you to the first of the current session.

Turn off the Message Scroll by removing the check beside **Messages** in the **WINDOW** menu. Turn it back on by replacing the check.



It is not intended for the message scroll to remain off as many important warnings and messages may appear here.

You can copy text from the Message Scroll area using standard Windows options (such as CTRL+INS to copy to the Clipboard and SHIFT+INS to insert from the Clipboard). This feature lets you copy an entry to the command line or even copy results from calculations into text within the project file.

The Terramodel Report Editor

In Terramodel, you can quickly and easily generate reports for your project, automatically formatted appropriately for the type of report you select. Edit your reports with the P3Pad Report Editor, adding notes and explanations where you want. Then, add graphics or change fonts and font styles to give your printed reports impact and a professional appearance.

The commands that send their output to the report editor include **Volumes, List,** and **Reports**. You can find more information about how to use the report editor in chapter 14, **Creating Reports**.

Chapter Three

How to Create, Save & Retrieve Projects

Creating a New Project

Before you can start entering data and designing your project, you must first create a new project file in which all of the data will be stored. Your file can reside in any folder, but we strongly advise that you keep all of your project files separate from the Terramodel executable files. This makes it easier to keep them organized and facilitates the process of updating Terramodel files from one version to the next while protecting your project files.



Consider organizing project files in subfolders that are arranged by job name.

To create a new project file:

- ▶ From the **FILE** menu, choose **New Project**.

The *New Project Name* dialog Box opens.

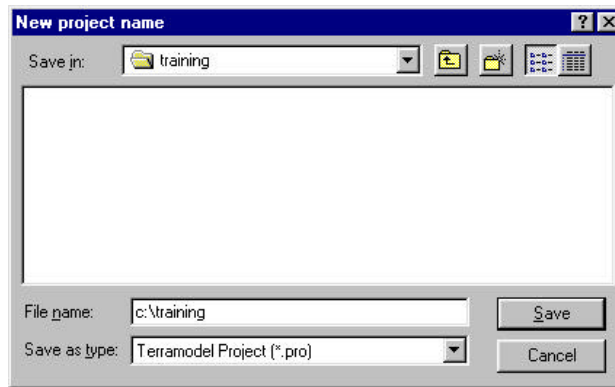


Figure 3-1 New Project Name Dialog Box

- ▶ Enter a name for the new file in the File Name text box. You can also select a file from the file list below to overwrite it and use its file name for the new file.
- ▶ Click OK to create the file.

Terramodel project files are created with a PRO file extension. When a file is saved the original file is given an extension of ~PR. In the event you must go back to the original file use the Windows file manager to assign a valid file name with the extension of PRO and then open the file from within Terramodel.

Your new project file can also be stored in the folder and drive of your choice.

To create the file in a different folder or drive:

- ▶ Change the folder under Save in by clicking a file folder icon next to the folder name of your choice. The folder is “open” when it has an open file folder icon next to it. If you want to create the file in a subfolder, you may have to open several directories until you see the name you want.
- ▶ Change the drive name by clicking the down arrow next to the Drives drop-down list box to display the drives that are available to you. Click the drive you want.

The new project file is first displayed from the plan view perspective. You will typically see a blank screen unless your file was created from a prototype that already contained objects. (For information on creating prototype files, see “Creating a Prototype Project File” later in this chapter.)

Building a Project File

When you build your project files, you should establish a layer naming convention and choose the colors and linetypes for the layers. The layering convention can be set up to accommodate categories of project files that share the same type of project.

You will also configure settings that appear under the **SETTINGS** menu, including input and display formats, precision settings and label abbreviations. Many of these settings remain constant from project to project, and can be established once in a prototype file. This prototype can then be used to begin all new projects. For more information on prototype files, see “Creating a Prototype Project File” later in this chapter.

Once you have prescribed your file settings, you can begin entering and manipulating data, calculating and displaying results, and printing and plotting. Below is a basic outline for building a project file. The steps will vary with your project requirements, and should be used as a guide in learning how to use Terramodel.

Establish Your Layering System

A layer is simply a property assigned to objects as they are created. After creation a layer provided a means of manipulating objects and identifying what objects are. Objects can be selected by layer and layers can be turned on and off. Layers provide a way to plot portions of the project file. In Terramodel Digital Terrain Models (DTMs) are created on a layer basis. Each layer can be a separate DTM.

Projects differ in their content and goals. Subdivision projects typically involve lot layouts, lot grading, road design, and storm and sanitary sewer planning and analysis. Roads require horizontal and vertical alignments, typical road sections, intersection layouts and storm sewer design. Each type of project involves different survey, design and drafting techniques.

Take inventory of what tasks will be performed, then decide how the objects you create will be organized among the layers. A system of naming layers that can be applied across all similar projects will help to keep your data organized. You will find that a well-planned project file helps the tasks flow more smoothly and facilitates making changes. Chapter 5, **Understanding Terramodel CAD**, contains a complete discussion of how to organize layers, as well as an example of a simple layering system.

Create a System of Colors and Linetypes for Your Layers

Assigning special colors and linetypes to represent groups of objects helps you to interpret data visually as you create it. Once you set up a coloring system, your eye becomes

accustomed to white points that represent original Digital Terrain Model (DTM) points and blue points that signify design points, for example. Likewise, sets that denote a road centerline and polylines that function as fence boundaries are easily recognizable when the appropriate linetypes have been applied.

Colors also provide control over plotted line weights. By mapping colors to specific pen numbers, you can plot lot lines with one pen weight and curbs with another. You can use up to 255 colors numbers in a project file, even though your graphics adapter may limit the colors that you can see.

Decide how you will organize your project by colors and linetypes, then continue to use that system from project to project. See chapter 5, **Understanding Terramodel CAD**, “Working With Colors” and “Working With Linetypes” to learn how to use the default color selection and linetype selection tools.

Set Your Input, Display and Precision Units

For your own ease of use, it is a tremendous advantage to incorporate a comfortable method of entering angle, distance and area data. From the **Units Settings** option of the **SETTINGS** menu (**UnitsSet** command), select a method of data entry that requires the fewest number of keystrokes, if possible. Also decide how you want your labeled angles, distances and areas displayed on the screen and in reports. See the Terramodel On-Line Help for more information on establishing project settings.

Set Up Your Views

When objects are created they have as a property a view mode. The view mode is used to determine how and when to display the object. Terramodel supports eight view modes, three of which are commonly used. The plan view mode is typically where most work is done and represents a Northing and Easting coordinate system. The profile view is the traditional engineers profile of an alignment. In this view mode the coordinate system is station and elevation. The sheet view mode is used to draft portions of the plan and profile in a sheet coordinate system of X and Y units. Having multiple view modes for data enables you to separate the design from the drafting data.

Using the **View Settings** option of the **SETTINGS** menu (**ViewSet** command), create view-appropriate coordinate label names; set a plotting scale and vertical exaggeration, if applicable. Once stored in a prototype file, these settings are available in any new project.

Import Your Original Data

Almost every project file will be built around existing information. Import your data from points files generated by data collectors or other software programs, from DXF/DWG files, or trace the information from a map using your digitizer. Whatever

your method, make sure that you are storing it on layers dedicated to this category of data.

See On-Line Help for information on the **Import** command.

Begin Your Survey, Design, Analysis or Drafting

The tools for this stage of project development vary widely depending on what you are hoping to accomplish. Every type of project, however, uses Terramodel's basic building blocks that include sets, points, polylines and text. Original DTMs and design models are created from *points* with elevations and *sets* that connect the points to help define the surface. *Polylines* are used to draw contours, planimetric features, sheet and title blocks and other lines that are not used for terrain modeling. *Attribute* and *text object* labeling and notes are drawn from text fonts and styles that you customize. Using these basic elements, your file can be as simple or complex as you need it to be.

Terramodel's menus feature commands that are grouped together by task. This method of arrangement should help you identify which commands to use and the order in which to use them.

A complete discussion of Terramodel objects and how they are created and edited is in chapter 4, *Understanding Terramodel Objects*. Digital terrain modeling is explained in chapter 7, *Creating a Digital Terrain Model*.

Analyze and Print Your Data

A good survey or design project adheres to standard guidelines for economic construction and operation. During and after the survey and design phase, use Terramodel's many calculation and report tools to assess your design.

For example, use:

- **Volumes** to compute earthwork.
- **Reports** to produce stakeout reports.
- **Pipesolv** to analyze sewer systems.
- **Hrout** to route a hydrograph through a pond.

Many of these commands send output to the report editor, where you can format, cut and paste, print, or export the data to another software application. Again, the tools you will use depend on precisely what you are doing.

You will find more information about creating and analyzing data for specific types of projects in the *Terramodel Instant Expert Guides*.

Create Dynaviews of Your Plot Sheets

Once your project has progressed to the stage where you are ready to plot, create a dynaview around the limits of each area that will be featured on a plotted sheet. Dynaviews let you establish and arrange your plot sheets (including plans, profiles, composite plan and profiles, cross sections and typical sections), and dynamically update these plot sheets to reflect any changes that you make to your project. Each sheet can have its own plot scale and rotation.

Place each project sheet in the sheet view in the order in which it will appear in your set of plans. Add typical sheet borders, north arrows and title blocks to each sheet as you create it, as well as construction notes where they apply. Chapter 4, **Understanding Terramodel Objects** and chapter 5, **Understanding Terramodel CAD** both cover how to use dynaviews in your projects.

Plot Your Drawings

You can plot to any of the printing or plotting devices supported by Windows using Terramodel's **Plot** command. You can also plot to files for use in other software programs. Chapter 16, **How to Plot in Terramodel**, provides information on the many options you have for plotting.

Saving Project Files

You can save your file at any time during editing. You can also preserve a file at a critical stage by saving it under another name. This feature is useful when you want to test different survey or design scenarios.

We recommend that you save and backup your project files **often**. If you experience a hard disk failure, there is no way to retrieve data except from a backup disk.

To save a project file:

- From the **FILE** menu, choose **Save Project**.

Your project file is saved and you are returned to the Command line.

To save a project file under another name:

- From the **FILE** menu, choose **Save Project As**.

The *Save Current Project As* dialog box opens.

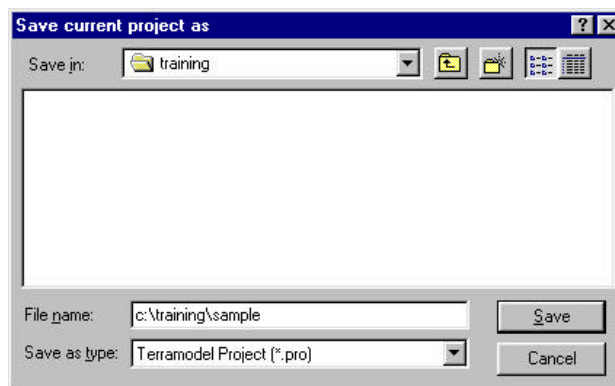


Figure 3-2 Save Current Project As Dialog Box

- Enter a name for the file in the File Name text box. To create the file in a different folder, change the folder name by clicking a file folder icon. Click Save to create the file. The new file name will become the current project file.

Closing Project Files

When you are ready to end your editing session, close your project file using these steps:

To close a project file:

- From the **FILE** menu, choose **Close Project**.

If you made any changes to your project file since it was last saved, the following dialog box will appear to let you save them before closing the file.

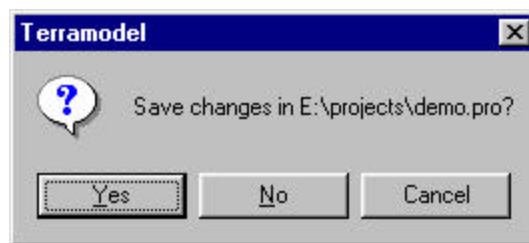


Figure 3-3 Save Changes in Project Dialog Box

- Click **Yes** to save the changes, **No** to discard the changes, or **Cancel** to keep the file open.

Retrieving Project Files

At the beginning of a Terramodel editing session you can start a new project file or open an existing one. If the file you wish to open is among the last documents used, open it quickly by choosing it from the bottom of the **FILE** menu. If the file has not been used recently, follow the steps below.

To retrieve an existing project file:

- From the **FILE** menu choose **Open Project**.

The *Open a Project* dialog box appears.

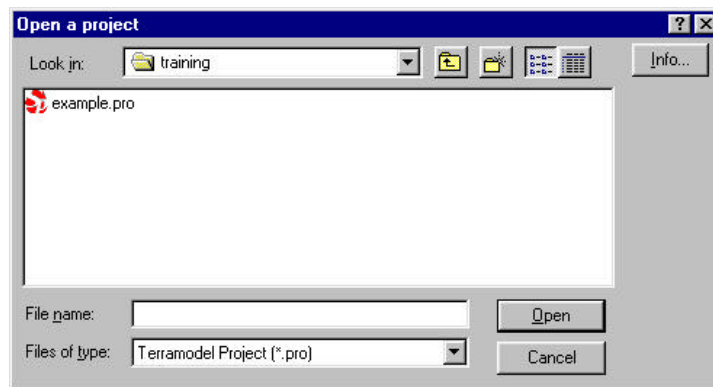


Figure 3-4 Open a Project Dialog Box

- ▶ Select a file from the list or, if you do not see the file you want to open, open another folder by clicking the appropriate file folder icon.
- ▶ Open the file you have selected by double-clicking the file name, or entering the name and clicking Open.
- ▶ The Info command button opens the *Terramodel Project File Info* dialog box, which displays information about the selected project highlighted.

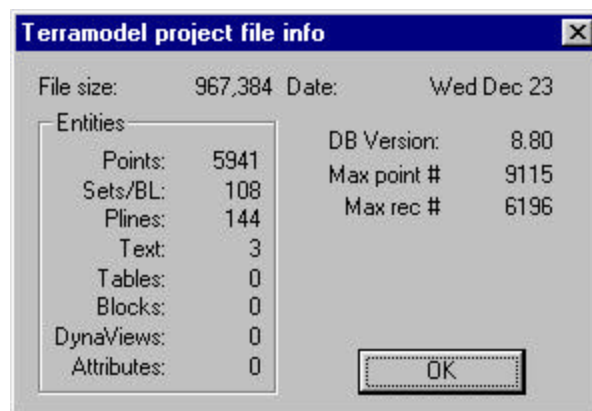


Figure 3-5 Terramodel Project File Info Dialog Box

Creating a Prototype Project File

Many of your project settings might actually be company standards that remain constant for certain categories of projects. These types of settings include layering conventions, colors, drafting settings, view configurations and units settings. Prototype files are used to store these settings in one location so that they can be recalled for all new projects. You can create multiple prototype files that can be used for different types of survey or design projects.

As you build your company standards you may want to add typical sheet borders, title blocks etc. to your prototype project. With a little preparation you can automate much of your basic drafting.

Terramodel includes a basic project file called TMODEL.PRO, which contains a few elementary settings, but no objects. Use this file as a basis for creating your own prototype file, then use *Save Project As* to save your settings under another file name (such as ROADPROT.PRO, or SITEPROT.PRO).

Once you have saved your settings, you can assign the new file as a prototype using the **Config System/System** option of the **FILE** menu. This option assigns the file as a prototype in the [User] section of the P3SERVER.INI file.

To create a prototype project:

- From the **FILE** menu, choose **Open Project**.

The *Open a Project* dialog box appears.

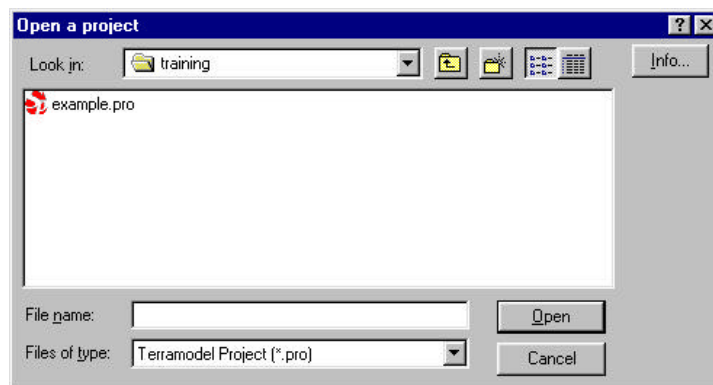


Figure 3-6 Open A Project Dialog Box

-
- ▶ Open the folder where Terramodel is stored by clicking the appropriate file folder icon.
 - ▶ Select TMODEL.PRO from the list of files. Open the file by double-clicking the file name where it appears.
 - ▶ Change the project settings to correspond to your own company standards. Examples of settings you might change:
 - Abbreviations.
 - Units settings.
 - Drafting settings.
 - View settings.
 - Layer settings.
 - Palettes and color mapping .
 - Text style settings.
 - ▶ Chapter one lists many of the critical settings you must establish to conform to your company standards and local terminology.
 - ▶ When you have completed your changes, open the **FILE** menu and choose **Save Project As**.
 - ▶ Type the name you want to give to your prototype project file, and click OK.
 - ▶ From the **FILE** menu, choose **Config System**, then **System**.
 - ▶ Enter the name of your prototype project file in the Prototype box.

COMMANDS TO CONSIDER FOR PROTOTYPE PROJECTS

AbbrevSet	LabelSeg (Settings)	Parking	ShapeMan
Alias	LabelSets (Settings)	PlanSet	SLL
CallOut (Load/New)	LabelTable	PlotSet	SnapSet
Carousel	LabelVal (Settings)	PlotterSet	StyleSet
ContourSet	LabPt	PointSet	Surface
DesignSet	LayerSet	PredArea (Settings)	UnitsSet
DisplaySet	Layout	RdDesignSet	ViewSet
DraftSet	LinetypeSet (Load)	RdsDTMSet	Zoom
GridSet	LinkSet	RdValDesCrit	AutoSave
Import (Pts Format)	MapPoints	RDX (Sheet Layout)	Coords
LabelGrid (Settings)	Materials	RoadGrid	ConvSet
LabelHal (Settings)	MeasUnit	ScaleGrid	Favorites
LabelPoint (Settings)	ObjSnap	SegEdit	System
LabelRoadHal	Palette	ShapeClass	ToolBox

Chapter Four

Understanding Terramodel Objects

Terramodel Objects

Terramodel uses six basic object types as the building blocks for a project file. These object types are:

- Points.
- Sets.
- Polylines.
- Text.
- Blocks.
- Tables.

Although different in nature, these object types share common properties. When you create objects of any type in Terramodel:

- Each object resides on a layer.

- Each object has a name.
- Each object has a visibility status of on or off.
- Each object has a color.
- Each object has a linetype.
- Each object has a record number, which is what **Terramodel** uses to keep track of the objects in the database.
- Each object can have a reference record number, which associates it with another object in the file.
- Each object can have attributes assigned to it, which allow you attach text that changes dynamically as the object changes, or use the attribute information in calculations.

A seventh object type, dynaview, is used to view and plot portions of a project as an image of the original. You can scale and/or rotate the dynaview as required. As you change the objects in the project file, the dynaview automatically updates itself to reflect the changes. Dynaviews are most often used to establish plot sheets and show enlarged details of a project area.

Each section in this chapter describes an object type and its particular properties, as well as the commands that you can use to create these objects. When you understand the basic object types and how they are used, you'll be amazed at how easy it is to build a project file.

Understanding Points

Points are the basis for most calculations in **Terramodel**. Field notes cannot be displayed on the screen, bearings, distances and areas cannot be labeled, and designs and volumes cannot be calculated unless points are first created.

A point object in **Terramodel** is equivalent to a surveyor's "point", or a point in space defined by a Northing, Easting, elevation and unique point number. A point can be a two-dimensional (2-D) or three-dimensional (3-D) element, whose attributes inside of a **Terramodel** file must include a Northing, Easting and point number, and may include an elevation and descriptor.

Rather than a point number, points may have a unique alphanumeric id (also called a point id). When creating a point and given the opportunity to specify the point number simply enter 16 characters or less with at least one non-numeric character. **Terramodel** actually maintains a unique point number for the point but "hides" it from the user and uses the point id.

Alphanumeric point ids may only be used only if they have been enabled by specifying a maximum number of alphapoints in the **System** command. A point with a point id rather than number requires an extra 20 bytes of storage. The searching for point ids is also slower than searching for point numbers. For these reasons point ids should be used sparingly.

You create points by importing surveying data from data collectors or other software applications, by tracing maps using digitizers, or by typing in the point information. Points are also created as a byproduct of both the creation of a new design, such as for a building site or a roadway, and changes to an existing survey or design.

In **Terramodel**, each point has a unique point number. The largest point number for a project is controlled by the value you enter for "Maximum number of objects" in the **Config System** option of the **FILE** menu.

How to Create Points

Points are created directly from information that you provide, or indirectly as a result of operations that certain commands perform. To store new points, you can enter north and east (y and x) coordinates or use your cursor to locate point positions on the screen.

Table 4-1 lists commands that you can use to store points.

Table 4-1 Point Commands

Command	Menu Location	Operation
Point	Draw/ Point/ Point	Stores points when you provide the coordinates. Optionally accepts point names and elevations for 3-D points.
Divide	Draw/ Point/ Divide Line	Creates points along a line at a specific interval. You can create points at stations, or divide the line into an even number of segments.
DtmGrid	DTM	Creates points along a grid, interpolating the elevations from a DTM.
DtmShot	DTM	Creates a point at the intersection of a line and a DTM.
Design	DTM	Intersects one DTM with another, creating new points where the two terrain models meet.
Merge	DTM	Merges two DTMs to form a composite DTM, creating new points and sets.
COGO commands	Cogo	These commands consist of line construction tools, street construction commands, and surveying tools.



For additional information on the commands listed in this section, see On-Line Help.

How to Select Points

Whenever Terramodel prompts you to locate a point, you can choose from two methods, as shown in Table 4-2:

Table 4-2 How To Select Points

Selection Option	Method
Graphical	Position the pick box over a point and click the left mouse button.
Point Number Entry	Type the point number or point id.

Understanding Sets

A set object is a special class of line that is defined by a list or set of point numbers that define straight segments and/or arcs segments. The locations of the ends of the segments are defined by the locations of the points that define the set. If a point in a set moves the set segment also moves. Sets have broad applications throughout **Terramodel**.

What Sets are Made Of

Sets are lines connected to points, and in fact cannot exist without the points that they connect. If you move or delete one or more points in a set, the set will be altered to connect the remaining points. If you delete all of the points, the set will also be deleted.

A single set can connect up to 65,000 points, and can connect points that occupy different layers. Sets can consist of line segments (the portion of the line between two points) or arc segments, which are defined by two arc points and one radius point. Sets are the only objects in **Terramodel** that can form arcs greater than 180°.

Open, Closed and Joined Sets

You can have *open* or *closed* sets. An open set has different beginning and end points, so that a listing to the report editor would look like this:

```
set points
Number of points → <5> 3 5 2 7 6
in the set
```

A set can also close back on itself so that, when listed, its beginning and ending points are the same. This type of *closed* set would be listed to the report editor like this:

```
set points
<5> 1 2 3 4 1
```


A set can be joined to another set that shares a common end point to form a single set. If a set containing points 1 and 2 were joined to a set containing points 2, 3 and 4, a listing to the report editor would look something like this:

```
set points
<4>1 2 3 4
```

Curved Set segments

A set segment can be used to define an arc of between 0° and 360°. In order to create an arc segment the distance between the PC and Radius (RP) and Radius and PT must match to within 0.001 units.

To define an arc of less than or = to 180°:

- While in the *Loc:* control of the **Set** command, click the PC point once with your mouse, click the radius point **twice**, then click the PT point. Clicking the radius point twice tells **Terramodel** the point is a radius point and to draw an arc from the PC to the PT. If the PC, RP, PT points form an arc of 180° the arc will be drawn clockwise. The small arc segment can be created by entering the point number of the PC, the point number of the radius point twice and the point number of the PT. The small arc segment can be created by entering the point number of the PC, a “-“ (minus sign) with the point number of the radius point and the point number of the PT.
- To enter multiple point numbers at once, click the **Pn** command button on the **Set** command line and enter the point numbers, separated by commas. Indicate the radius point by preceding it with a minus (-) sign (ex. 2,-502,10). You may also enter the radius point twice. If there are back to back curves the adjoining PC/PT point must only be entered once. You may also enter point numbers one at a time in the *Loc.* control.

To define an arc of greater than or = to 180°:

- While in the *Loc:* control of the **Set** command, click the PC point once with your mouse, click the radius point **three** times, then click the PT point. If the PC, RP, PT points form an arc of 180° the arc will be drawn counter-clockwise. The large arc segment can be created by entering the point number of the PC, the point number of the radius point three times and the point number of the PT. The large arc segment can be created by entering the point number of the PC, a “+“ (plus sign) with the point number of the radius point and the point number of the PT.

- To enter multiple point numbers at once, click the **Pn** command button on the **Set** command line and enter the point numbers, separated by commas. Indicate the radius point by preceding it with a plus (+) sign (ex. 31,+68,95). You may also enter the radius point three times. If there are back to back curves the adjoining PC/PT point must only be entered once. You may also enter point numbers one at a time in the *Loc.* control.

You can also use the **Traverse**, **Fillet** and **Scurve** commands to create arcs in set objects.

What Sets are Used For

Surveying

When you import survey information from field notes, **Terramodel** reconstructs and stores it in your project file using points and sets. These set lines can be labeled with bearings and distances, and the areas of closed sets. As you make changes to your project, the labels attached to the sets will be updated automatically.

The **FieldD** command in **Terramodel** adjusts and balances survey field notes and transfers the information into a project file as sets and points. For more information on **FieldD**, see On-Line Help.

Designing

When sets connect points around the perimeter of a DTM, they can be used by the **Design** command to project one DTM onto another. **Terramodel** projects your designated sideslopes from these perimeter sets to intersect the other DTM, and creates new sets along these projections. The tops and toes of slopes are also traced with new sets. All of these sets help to model the surface and enhance the accuracy of subsequent calculations.

Breaklines

Breaklines are lines that control how a DTM is modeled. Connected to 3-D points, they affect contours, volumes calculations and the direction of ground slope. Set objects function as breaklines in **Terramodel**, when they connect 3-D points that occupy the same DTM and the set is on the same layer as the points.

Terramodel can automatically add sets between points having the same point names, using the **Mappoints** command.

For more information on how sets are used as breaklines, see chapter 7, **Creating a Digital Terrain Model**, in this guide.

Smoothness of Contour Lines

Sets can also affect the smoothness of contour lines that pass through them. When you apply a smoothness factor to a set, you can make the contours smooth, have slight bends, or sharp bends, without affecting the DTM surface.

To learn how to apply smoothness factors to sets, see the **SetSmooth** command in the On-Line Help.

Boundary or Survey lines

Sets are used in subdivision layout and design to represent property boundaries, lots, and street right-of-ways. These types of sets can display bearings, distances, curve properties, and area labels with attribute text that changes dynamically as the layout of the subdivision, lots or street changes.

Horizontal Alignments

Sets can be used to describe the center line of a roadway. When you use them in this fashion, **Terramodel** can label the alignments with text objects that display stationing, curve points and horizontal curve data.

If the alignment contains spiral or compound curves the polylines must be used rather than sets.

Sewer Design

A set can also be used to define a sewer pipe that the Sewer Module can design and analyze. **Terramodel** can assign pipe attributes to it, such as the pipe size, shape and material. Once **Terramodel** has analyzed the pipe and computed flow configurations, the calculated pipe properties can also be assigned to the pipe as attributes and **Terramodel** can plot and label the pipe in the profile view. You can also show 3-D sets in profile as pipes that cross an alignment or another pipe system.

How to Create Sets

Like points, sets can be created directly from information that you provide, or indirectly as a result of operations that certain commands perform. When you enter sets directly you can connect existing points with a set line, or you can pick positions in the **Terramodel** graphics screen. For the latter method, **Terramodel** will create new points at the positions you pick and connect the new points with set lines.

The table below lists commands that you can use to create and edit sets.

Table 4-3 Set Commands

Command	Menu Location	Operation
Set	Draw/Set	Creates and draws new sets and set points, connects new set lines with existing points.
Breakline	Draw/Set	Creates and edits set lines that connect 3-D points on the same layer.
Pad	Draw/Set	Creates a rectangular, closed set and assigns default sideslopes.
Autoset	Draw/Set	Creates a set in a clockwise direction from selected points.
Set Smooth	Modify	Assigns or changes the smoothness attributes of selected sets.
DtmEdge	DTM	Creates a set line around the edge of a DTM.
DtmDrain	DTM	Creates set lines around the drainage areas of a DTM.
Design	DTM	Intersects one DTM with another, creating new sets where the two terrain models meet.
Merge	DTM	Merges two DTMs to form a composite DTM, creating new sets and points.

Command	Menu Location	Operation
Edit commands	Edit	Changes the properties of selected sets, including breaking and joining sets, converting them to and from polylines, trimming and extending sets, and creating offset set lines.
Design settings	Settings	Applies smoothness to new sets, assigns cut and fill slopes to design sets.
Curves commands	Cogo	Construct set curves using best-fit curve and other curve solutions.
Lines commands	Cogo	Construct set lines using bearing-bearing, bearing-distance and distance-distance intersections.
Streets commands	Cogo	Construct simple roads and Cul De Sacs. Perform basic road design.
Traverse	Cogo	Creates set lines and curves given bearings, distances and other surveying information.
Pre-determined area	Cogo	Computes and constructs set lines to represent lots, given specified lot areas.
FieldD	Cogo	Creates points and sets from survey field notes.

How to Select Sets

Whenever Terramodel prompts you to select a set, position the pick box over a set and click the left mouse button.

Understanding Polylines

In **Terramodel**, a polyline object is defined by a series of vertices and curve parameters drawn to coordinate positions. Polylines are primarily used for contour lines (so they can be splined or smoothed), horizontal alignments, vertical profiles and general linework.

What a Polyline Consists Of

Polylines can consist of straight line segments, arcs, splines, or parabolic and spiral curves. Unlike sets, they do not need point objects to define them and in fact cannot connect to points. Polylines are defined by the location of their vertices (also known as control points) and their curve parameters, if any.

Polylines can have circular, spiral, overhauser (also known as Catmull-Rom) or parabolic curves. Each type of curve has special characteristics and uses. Circular and spiral curves are used for curves in plan views, while parabolic curves represent those typically found in vertical alignments.

Types of Polylines You Can Create

Polylines are two-dimensional and may have a single elevation. A typical usage of this is a contour line.

Like sets, polylines can be open or closed. When you list a closed polyline, its beginning and ending control points (vertices) will have different numbers, although their coordinates will be identical.

Polylines are the only type of line which can be splined (smoothed), and are the only **Terramodel** object type that is used for contour lines. You can create splined or unsplined contours, and change the amount of splining to suit your needs with the **View Resolution Segments per Spline** option of **Display Settings** (**SETTINGS** menu, or **DisplaySet** command). **Terramodel** uses the Cubic B-Spline or overhauser (also known as Catmull-Rom) method of splining, also used by AutoCAD and other CAD software. Examples of these spline types are shown in Chapter 7, **Creating a Digital Terrain Model**.

What Polylines are Used For

Contouring and Planimetric Features

Terramodel uses polyline objects to represent the contours generated with the **Contour** command. Polylines are also used for representing planimetric features (such as sidewalks, building lines, etc.). Since they are not three-dimensional and cannot affect the links on a DTM, they are not used on layers that contain DTM information or that will be used for terrain modeling.

Horizontal and Vertical Alignments

Although you can use both sets and polylines to draw horizontal alignments, Terramodel uses only polylines to create vertical profiles along DTMs with the **Profile** command. And while you are in the profile view, you can use only polylines to design new profiles. Polylines have the ability to support the curve types needed for horizontal and vertical alignments.

Dynaviews

Polylines are also used to define the limits, or boundary, of dynaviews. Use them to define the limits of what should be seen through a dynaview in the sheet view or other view modes. For additional information about dynaviews see the **Dynaview** command in the On-Line Help.

Plotboxes

Use plotboxes to outline the limits of an area in your project file that you want to plot. The **Plotbox** command lets you create a closed polyline boundary in terms of actual sheet heights and widths. You can then reference this boundary when you create a dynaview. For more information about how plotboxes are created, see the **Plotbox** command in the On-Line Help.

Borders and Title Blocks

You can use polylines to draw the title blocks and borders that frame plot sheets. They can also be used to create blocks (like a north arrow block) that can be accessed by

multiple **Terramodel** project files. For more information on how blocks are created and used, see the block command in the On-Line Help system.

Cross Section Locations

Terramodel's Xline command creates polylines that denote the width and location of cross sections. These xlines are used to interpolate cross section points that are stored in ASCII files or used by the ROADS module for roadway design. For more information on using the **Xline** command, see the On-Line Help system.

Grid Lines in Dynaviews

The grids that are displayed in the plan and profile views are not **Terramodel** objects, and cannot be viewed through a dynaview. You can create grid lines or ticks from polylines that can be seen through dynaviews using the **GridMake** command.

How to Create Polylines

Terramodel includes several commands specifically designed to create or edit polylines. The table below lists them:

Table 4-4 Polyline Commands

Command	Menu Location	Operation
Pline	Draw/Pline	Creates a polyline object.
Box	Draw/Pline	Creates a four-sided polyline.
Circle	Draw/Pline	Creates a polyline emulating a circle.
Plotbox	Draw/Pline	Creates a rectangular polyline used to define the size and contents of a plot sheet.
Xline	Draw/Pline	Creates polylines that denote the width and location of cross sections.

Command	Menu Location	Operation
Grid	Draw/Pline	Creates grid lines and ticks that can be viewed through a dynaview.
Generate contours	DTM	Links and contours a DTM.
Create profile	DTM	Draws a profile of a DTM in the profile view along a selected alignment.
Contour settings	Settings	Establishes the default settings the Contour command uses to create contour lines.
Display settings	Settings	The View Resolution for Splines option establishes the degree of smoothing for splined polylines.
Edit commands	Edit	Contains options for breaking and joining polylines, converting them to sets, adding curves, creating a matrix, offsetting polylines, and trimming and extending.
Create parking stripes	Cogo	Creates polylines to represent parking stalls.

How to Select Polylines

Whenever Terramodel prompts you to select a polyline, position the pick box over a polyline and click the left mouse button.

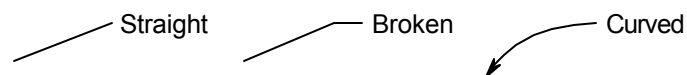
Understanding Text

Terramodel uses two basic types of text: normal *text objects* and *attribute text*. They are each described briefly below.

Text Objects

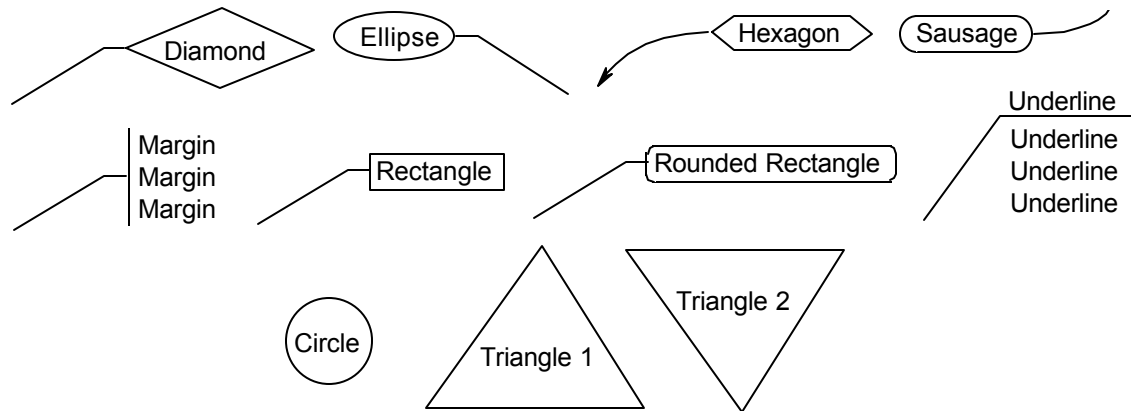
Text objects contain the textual content that you place in a project file. A text object might contain a single character, a word, a single line of text, or multiple lines of text. A multi-line text object can have its lines left, center, or right justified, and even though it may contain an entire paragraph, it is only one object.

A text object can optionally be shown with a *leader line* that points to its *subject*, i.e., the object being labeled. Either of three different styles of leader lines are available as shown below.



A leader line is often used with what is generally known as a *callout*, i.e., text that labels an object. You can assign any supported linetype to the leader line, thereby employing various arrowheads if you wish. As you move the text object, the leader line will remain pointing to the subject of the callout. As you move an object such as a point or block, if there are text objects using leader lines for which it is the designated subject, those leader lines will continue to point to that subject after it is moved. As we'll discuss below, in addition to pointing to and tracking a subject, a text object can automatically display numerous of the subject's attributes and automatically update its displayed content as those attributes are edited.

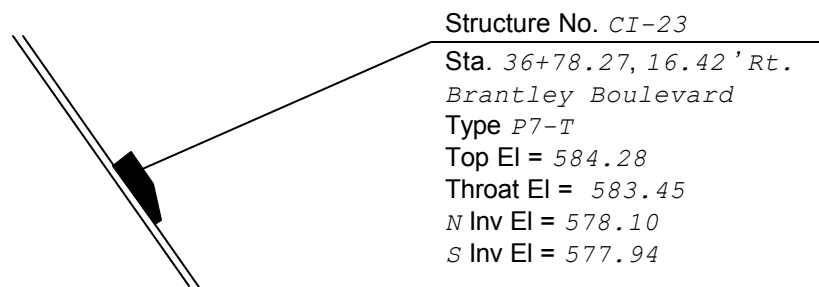
A text object can optionally be shown with a *border line*, which can be used alone, or in combination with either of the supported leader lines. Twenty different styles of borders are supported, a few of which are shown below. Some of these border styles are shown in combination with leader lines in order to illustrate how they work together, but they can be used without the leader line if you wish. With the exception of the *Margin* and *Underline* styles, they will expand to enclose the text regardless of its size. Those two particular border styles do not enclose the text, but do expand as needed to accommodate its height and length, respectively.



The leader and border lines are not separate line objects. They are actually a part of the text object that simply displays with it. To select the text, you click on the text itself, not the leader line or border. Since the leader line and border are not objects, they can't be selected. When you move or rotate text that employs a leader line, the leader line will remain pointing where it's supposed to. When you move the text to the other side of that point, the leader line will flip to the opposite side of the text.

One of the more powerful features of **Terramodel** is its *smart text*. Smart text uses *Embedded Attribute Text* (EAT) codes, which can, among other things, display various attributes of the text itself, its parent object, the subject of a callout, various project file data, project variables, or user defined object attributes. Some of the attributes that can be displayed include an object's coordinates, its station and offset with respect to an alignment, its name, its layer, its point number, and other object attributes as assigned by design programs such as contained in the Sewer Design and Analysis module. Text objects can also include references to points, which when combined with smart text codes can display the bearing and/or distance between those points. This type of text, referred to as a segment label, can only be created with the **LabelSeg** command. The text object might be composed entirely of an EAT code, or there may be one or more EAT code substrings embedded within a text object.

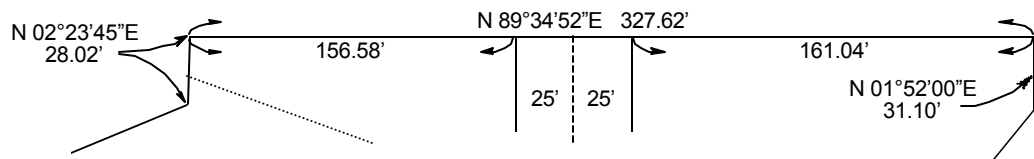
Using these features, you'll be able to create sophisticated callouts such as that shown below.



In this example, which labels a curb inlet, the text shown in italics and a different typeface is that which is displayed as a result of an EAT code. In this instance we've created EAT codes that display the name of the subject curb inlet object, its station and offset with respect to the roadway on which it lies, that roadway's name, the curb inlet's type attribute, its top and throat elevation, and the invert elevation and direction attributes of the connecting storm sewers. All of this data is automatically extracted from the curb inlet object. More importantly, if any of this data changes, as a result of the inlet's relocation or redesign, the callout will automatically update to reflect the current status of that design. As you can see, once you place a callout using leader lines and EAT codes, you're essentially finished with that text. No longer will you have to continually edit these callouts as the inevitable design changes take place, nor will you risk forgetting to do so. If the subject moves, the leader line will follow it and its coordinates or station and offset will update accordingly. If the name of the roadway is changed, every text object referencing that name updates. If the storm sewer system design must be changed, the text which references the design results will update to reflect that change. The possibilities of what you can accomplish with smart text are limitless.

In order to facilitate the labeling of commonly encountered objects and site features, you can create libraries of predefined *callout styles*. A callout style contains the text and EAT codes necessary to label a particular type of object, such as the curb inlet in the above example. In addition, it defines the leader line and text border styles to be used, if any. To label the curb inlet, you simply select the appropriate callout style, and click on it. The label is placed in accordance with the callout style definition.

It is a common convention in the preparation of survey drawings to specifically and clearly mark the points at the end of a line segment being labeled, in certain cases where this would not otherwise be clear. This is often accomplished using small curved arrows projecting from the end points toward the label. These arrows are referred to by some as *crow's feet*. Crow's feet are used to fully clarify the line segment end points to which a label refers. An example is shown below.



Terramodel can draw these crow's feet for you as a special form of leader line. They are therefore an integral part of the segment label text object. They dynamically mark the location of the specific points that the segment label refers to, moving in response to a

relocation of the referenced points, as the label text updates to indicate the revised geometry.

When creating segment labels, the text is typically placed alongside the line segment, as shown along the top boundary in the above illustration. This placement is performed automatically by **Terramodel** when creating segment labels using the **LabelSeg** command's *Auto mode*. The label remains registered to the line segment as its end points are relocated. In some cases, the line segment may be too short to accommodate the label. In these cases you can opt to create an entry within a line or curve table, placing just a reference to the table entry by the line segment, or you can just pull the label away from the line by employing the *User mode*. In this case, you can configure the segment label style to automatically place a leader line pointing to the midpoint of the line segment, as shown on the right side of the above illustration. When in such a case, you need to explicitly mark the end points of the line segment being labeled, instead of using the normal crow's feet to do so, **Terramodel** will employ two separate leader lines, as shown in the example on the left side of the illustration. As the text object is moved, these leader lines continue to point to the line segment midpoint or end points as the case may be.

You can also use a text object to display the contents of an external ASCII text file in your drawing. To do so you simply create a text object of the following form, where *filename* is the name of the external file.

\$File:*filename*

Terramodel will retrieve the designated text file and display its contents at the location of the above text object, using the current text style. Typically you'll want to use a text style with Top/Left justification when doing this, so that the position at which you place the text becomes the upper left corner of the resulting text display. Textual content displayed in this manner can not be edited from within **Terramodel**. You can edit the external text file and save it. **Terramodel** will then display those edits the next time you issue the **Redraw** command.

Text objects are currently limited to a total of 3095 characters.

Text objects, and attribute text as discussed below, are created based on the current text metric settings. These current settings can be edited, and can be set to those of a predetermined *text style*. A text style specifies a complete set of text metrics and CAD properties that you may wish to employ routinely. These text metrics and CAD properties are also properties of the individual text object, and can therefore be edited as desired at any time. The current text metric settings just provide the initial default settings so you don't have to specify each of them each time you create a text object.

You can select from several text orientation settings that control how text is displayed as it is rotated, or displayed within rotated dynaviews. When your zoom setting is such that attribute text or text objects are displayed at less than the number of pixels

specified by the *Text Box Size* option of the **DisplaySet** command, **Terramodel** draws surrounding boxes rather than the actual text, in order to speed the screen redraw.

For more information about text styles, fonts, EAT codes, displaying external text files, and other CAD properties of text, see chapter 5, **Understanding Terramodel CAD**, “Working with Text and Text Styles”.

Attribute Text

Attribute text is the displayed result of internal computations performed by **Terramodel** and is not a database object. It is created when you:

- Label closed sets with areas.
- Label set segments with a line or curve table.
- Label points with symbols and block definitions.

Attribute text requires no additional memory since it is not an object. **Terramodel** simply computes the properties of the line, arc, or closed set, and displays or plots them. Attribute text will update to reflect the current geometry if you move the points of a segment which is labeled in a table, if you move the points of a closed set for which the area is labeled, or if you move a labeled point.

Attribute text cannot be selected or edited using any of the text edit or **MODIFY** options. Attribute text changes only when the computed geometric properties change as a result of modifications made to the position of points or sets in your project, or when you make changes to the default settings within each of the **Label** commands that control how attribute labels appear.

How to Create and Edit Text

Terramodel includes several commands specifically designed to enter or edit attribute text and text objects.

Table 4-5 Text Commands

Command	Menu Location	Operation
Single line text	Draw/Text	Creates a simple single-line text object.
Text	Draw/Text	Creates a text object with options for all of the text properties.
Callout text	Draw/Text	Creates a text object from a predefined callout style. Allows you to create and edit callout styles.
From an ASCII file	Draw/Text	Imports ASCII text files into the drawing as text objects.
Text along	Draw/Text	Creates a text object along a selected set or polyline. The text will bend to follow the line.
Edit object	Edit	Edits the textual content of a text object. Assigns leader and border lines and their properties. Edits the CAD properties, and text metrics for the selected text object.
Label commands	Draft	Create attribute text labels for points, tables, closed sets. Create text objects for grid, set segments, horizontal and vertical alignment labels.
Modify text commands	Modify	Allows you to change the height, aspect ratio, style, scale, alignment and rotation, etc. of one or more text objects.

Command	Menu Location	Operation
Draft settings	Settings	Sets drafting default settings for attribute text and text objects.
Abbreviations	Settings	Establishes the prefixes for attribute text and text object labels.
Text style settings	Settings	Creates new and sets current text styles for text objects.
View settings	Settings	Sets the drawing scale used to size text and plotboxes.
CrowFoot	Keyboard Only	Toggles on and off, the crow's feet arrows that mark the ends of a line segment being dimensioned in a survey drawing.

How to Select Text

Whenever **Terramodel** prompts you to select a text object, position the pick box over the text and click the left mouse button. In selecting text which incorporates a leader line or border, you must click on the text itself. The leader line and border can not be selected, since they are not actually line objects.

Understanding Blocks

Blocks group one or more objects together to create a block definition. Block definitions can be inserted in project files multiple times as unique block objects, with each object having its own insertion point; reference to a point object; X, Y, Z scales; and rotation angle. You can keep a block definition inside one project file, or create it externally to use in multiple project files. External block definitions are automatically updated when their components change, but they must reside in the **Terramodel** search path (TSP) in order to be accessed. When external block definitions cannot be found, **Terramodel** displays an error message as the project file is loaded, and the block is not displayed.

What Blocks are Made Of

Blocks can consist of points, sets, polylines, dynaviews, and text. They can contain only one object, or many objects. Even very large blocks conserve your program memory because each object contained in a placed block retains only the properties required for its use as a block component, and therefore requires less memory than the original definition of the same object.



You can create a block from a dynaview, but the block will not be updated each time the dynaview changes.

What Blocks are Used For

A primary use for blocks is hatch patterning. A pattern is a series of repeated lines that are clipped to a boundary. The **Hatch** command uses block definitions to create a pattern, which is then inserted as a single block object inside of and clipped to a selected boundary. The advantage of using blocks for hatch patterns is that all of the block components become a single object that can be erased or moved as a group.

Blocks can also be created to label points with coordinates, elevations, and other information. A block gives you the freedom of labeling what you want, placing the labels in any order, and including other **Terramodel** objects such as polylines for leader arrows and boxes. This allows you to customize your point labels.

Another use for blocks is creating multiple block objects from a single definition. An example of this would be creating a single block definition for a title block and border sheet that can be used for all of your plot sheets. Other uses include symbols that can be created once and then used over and over in project files.

How to Create Blocks

The **Terramodel** commands for creating and manipulating blocks are shown in the table below:

Table 4-6 Block Commands

Command	Menu Location	Operation
Block	Draw	Sets block defaults, creates and edits blocks, purges unused blocks, inserts blocks into projects.
Hatch	Draw	Creates hatch patterns and places them as blocks
Explode	Edit	Explodes placed blocks into their individual components.

Terramodel includes several useful block definitions of title blocks and north arrows to save you time.

Exploding Blocks

Blocks that are exploded into their individual components follow these guidelines:

- Points contained in the blocks are assigned new point numbers.
- All lines within blocks are converted into polylines.

- Some minor loss of accuracy in coordinate values may occur from the conversions. This loss will typically be less than 1×10^{-10} units, but can change a point having an undefined elevation (1×10^{-29}) to one having a very small elevation value as a result of the rounding applied.
- Exploded blocks do not retain the attributes of the original objects. If a block is created from a point labeled with a point number, when the block is exploded the new point will have a different point number, and the point number label will be a text object rather than an attribute of the point.

For additional information regarding blocks see the **Block** command in the On-Line Help system.

Understanding Horizontal and Vertical Alignments

Horizontal and vertical alignments are two special uses of set and polyline objects.

Horizontal Alignments

Horizontal alignments (HALs) are created in the plan view using sets or polylines. They consist of straight line tangent segments and can also have circular or spiral arcs. After you have entered them, you can label their properties using the **LabelHAL** command.

Generally, HALs are used as survey baselines and for roadway and subdivision street design. You can also enhance their capabilities with the ROADS module to develop a full-fledged highway design. By registering a HAL in the ROAD's HAL Manager, you can apply complex template and subgrade configurations, incorporate station equations, and use it to define horizontal template transitions.

Vertical Alignments

Vertical alignments (VALs) are always created in the profile view. VALs are only entered using polylines, and can contain straight tangent sections, as well as parabolic curves. You can enter the control points (PVIs) for a VAL in the profile view, or **Terramodel** can interpolate and draw vertical profiles from a DTM. Your vertical alignments can profile either existing or design surfaces. Once entered, you can label the vertical alignment properties using the **LabelVAL** command.

VALs are also used in ROADS to expand your roadway design capabilities. Registered VALs allow you to define existing and proposed vertical alignments, as well as transitions in templates and subgrades based on a given vertical alignment.

How HALs and VALs are Used Together

You can use HALs and VALs along with simple template information to develop basic roadway designs, like projecting roadway sideslopes to intersect a DTM using the **SmpRoad** command, or interpolating roadway and cut/fill points from a DTM surface with the **Slice** command. They also work together in the ROADS module to build complex roads and highways.

When **Terramodel** creates vertical profiles with the **Profile** command, it automatically associates (or references) the new vertical profile with the horizontal alignment you selected during the command. Other objects can also be referenced to that HAL. If Auto Reference is on in the **ViewSet** command, only the profile and other objects referenced to the HAL will be displayed in the profile view. This object referencing feature organizes your project file so that you only see and work on information in the profile view that is pertinent to a particular alignment. For more information on object referencing, see “Referencing Objects to Other Objects” on page 124.

How to Create HALS and VALS

Table 4-7 shows the commands you can use to create or manipulate horizontal and vertical alignments.

Table 4-7 HAL and VAL Commands

Command	Menu Location	Operation
Set	Draw/Set	Creates points and set lines for a HAL.
Pline	Draw/Pline/ Line	Creates a polyline for a HAL or VAL.
Curve	Draw/Arc/ Curve	Adds arcs (circular) or spiral curves to HALS, vertical (parabolic) curves to VALs.
LabelHAL	Draft	Labels HALS with text objects to display alignment and curve properties.
LabelVAL	Draft	Label VALs with text objects to display vertical properties.
Layout	Cogo/Streets	Creates offset HALS, intersections and Cul De Sacs.
Slice	Cogo/Streets	Interpolates and creates points from a DTM given a HAL, VAL and simple template information.
SmpRoad	Cogo/Streets	Creates template points and sideslope intersections with a DTM given a HAL, VAL and template.
HALManager	Road	Registers a horizontal alignment with ROADS, defines station equations and alignment offsets.

Command	Menu Location	Operation
LabelRoadHal	Road	Labels a Roads horizontal alignment with text objects to display alignment and curve properties.
LabelRoadVal	Road	Labels a Roads vertical alignment with text objects to display alignment and curve properties.
ValEdit	Road	Edits the vertices of a vertical alignment
VALManager	Road	Registers a vertical alignment with ROADS, defines alignment offsets.

Understanding Dynaviews

Dynaviews are used in **Terramodel** to display a portion of your design in another location. A typical application is to create a dynaview around a portion of the plan view, then create another dynaview around the related portion of the profile view and to combine these onto a plan and profile sheet in the sheet view. Another usage is to make an enlarged view of a detailed area. The advantage of using dynaviews is that they provide a dynamic view of the design and not a copy, saving memory and preventing transposition errors. When the design data is revised, the images on the sheets within dynaviews that look into that data are automatically revised.

How to Create Dynaviews

You establish the outline (shape) and size of a dynaview by creating a polyline in the plan, profile, sheet or xsect view around the objects you want to be shown through the dynaview. This might typically done with the **Box** or **PlotBox** command to create a rectangular polyline, or with the **Pline** command to create a plotbox of an irregular closed shape. Once you create the polyline surrounding the objects to be shown in the dynaview, you can use the **Dynaview** command to create the dynaview object itself. In doing so you select the polyline boundary and specify a view, location, scale, and rotation angle for the dynaview object. The dynaview is referenced to the plotbox, making it the dynaview's "parent" object. The dynaview displays the objects that are located within its parent's closed boundary. What is shown in the dynaview is dependent upon the location and shape of the parent, and if the parent is moved the dynaview is automatically updated. Dynaviews can be placed in any **Terramodel** view mode and its parent can be from any view mode, so that plan, profile, and cross-section views can be combined on one sheet.

Dynaviews are generally intended to be referenced to rectangular boundaries, although you can use non-rectangular dynaviews with some limitations:

- **Terramodel** generally draws non-rectangular dynaviews more slowly because of the more complicated clipping required.
- Spiral and parabolic vertical curves are not clipped to non-rectangular dynaviews.
- Line patterns will start at the clipped boundaries and hence the dynaview may not show a true picture of the original line pattern.

How Objects are Displayed through a Dynaview

Your dynaview image will look almost identical to the original objects. Typically, the images of any objects (except text) that cross the polyline boundary will be clipped to the boundary when you look at them through the dynaview. There is an exception, however, as described below.

The images of text are not clipped to the boundary. If the insertion point of the text lies within the dynaview boundary **Terramodel** draws the entire text string, even if it extends beyond the limits of the dynaview. This is done so that a portion of valuable notes is not accidentally dropped, or so that an “8” does not get clipped to look like a “3”. If the text insertion point falls outside of the dynaview limits, **Terramodel** omits the text in the dynaview.

The exception noted above is that objects that are children of the dynaview’s boundary will be displayed with the dynaview, even if they lie outside of its boundary. If the insertion point of a text object falls outside the dynaview boundary you can force the text to be displayed in the dynaview by referencing the text to the dynaview boundary.

Text objects that are assigned an orientation setting of legible, will automatically flip as needed when displayed within a rotated dynaview, to read from the bottom or right side of the sheet. Text objects that are assigned a fixed orientation will remain oriented as they are drawn in the target viewing mode, even if the dynaview is rotated.

Line patterns and text in a dynaview are scaled according to the dynaview scale. In rectangular dynaviews the line patterns will match the original.

Dynaviews do not display the grid from the target view because you would be limited to a single grid type and spacing for every dynaview that has a boundary in the same view. After you select a grid type and spacing, you can use the **GridMake** command (**DRAW/Pline/Grid** from the menu bar) to create polylines from the grid lines that can then be seen through a dynaview.

If your dynaview becomes quite large and complex, you can speed screen regeneration times by selecting the **Quick Dyna** option of **SETTINGS/Display Settings**. This setting causes **Terramodel** to show the dynaview outline only, without displaying the images of the objects inside of the boundary. This is useful when positioning multiple dynaviews. For every dynaview **Terramodel** must traverse through the project file and determine what items must be displayed. Although this process is remarkably fast you should not use dynaviews to show one or two details. An example of an improper use would be to have one for the P. E. stamp, one for the title block, one for the legend, one for the North arrow, etc. for each sheet of multiple sheets. In this case blocks would be better suited or use a single dynaview to draw all of the standard items for a sheet.

In order to be able to completely control what is displayed in multiple dynaviews, each dynaview can have a named layer list as a property. The **LListSet** command can be used

to create a named list of layers. When this layer list is assigned to a dynaview it controls the layers that will be visible in the dynaview. Objects on layers not in the list will not be visible in the dynaview. Objects on layers that are members of the layer list will be visible in the dynaview, even if those layers are set to invisible in the target view. This feature is used in the ASAP module so that an entire set of plans can be created using dynaviews without having to change layer visibility for each sheet to be plotted. Layer lists assure that your sheets are always plotted with the appropriate content, regardless of the current visual editing state of your data.

In addition to controlling the content of a dynaview, an associated layer list can be used to control the prominence with which objects are plotted. Objects on a given layer can be plotted in the foreground in one dynaview, while those same objects are plotted in the background, i.e. screened perhaps, in another. For more information on layer lists and variable object prominence,, see chapter 5, **Understanding Terramodel CAD**, “Working with Layer Lists”.

Dynaviews and Object Referencing

Dynaviews make viewing and drafting portions of the design data simple. Explaining the rules that are used seem complicated but because of the natural flow of the design and drafting process the user is not required to know all of the details. The rules for displaying objects when using multiple dynaviews are best understood with the metaphor of ancestors and dependents; a family tree structure.

The original boundary is a parent. The dynaview created from the parent is a child. The reference record of the dynaview is the record number of the parent. Multiple children can be created from the same parent, but each child can have only one parent (reference record). This simple system enables **Terramodel** to determine if any object belongs to a family and hence when it should be drawn.

Objects referenced to the parent (the dynaview’s boundary pline) are displayed, regardless of whether they fall within the limits of the parent. Reference any object to the parent of a dynaview and it gets plotted with the dynaview. All other objects are clipped to the limits of the dynaview’s boundary pline.

If the object has a parent that is not a part of the tree the user has the option to automatically turn these off or not. This is controlled by the “Auto off” setting when using the **Edit** command and editing a dynaview. In **Terramodel** versions prior to 9.10 this setting was always enabled and is typically the proper setting for a dynaview with parents in the profile view mode. One application where this option is needed is when using the **Layout** command. In this command the offset lines are referenced to the centerline of each road, however the centerline is typically not referenced. With “Auto off” enabled the offset lines would not be displayed in a dynaview, when disabled they

would be displayed. The user may optionally reference the dynaview and centerlines to the same object and then they would be displayed.

Terramodel determines the great-grand parents of the dynaview, and plots all objects, with a reference, that are descendants of the great-grandparent, with the exception of objects, and their descendants, that are children of other dynaviews.

Although the process sounds complex the effect is that you can have multiple overlapping dynaviews and each view will only show the objects that it is suppose to. This process works as a natural by product of creating horizontal alignments, vertical alignments, labeling the alignments and creating dynaviews around portions of the alignments to create fully annotated plan and profile sheets.

Changing a Dynaview

You cannot select the images within a dynaview to change them—only the original objects can be changed. If you want to move or otherwise edit the characteristics (location, scale, rotation) of a dynaview, the **Search** status for dynaviews must be on and using the **Edit object** command, you must select its origin, or the starting point of the polyline boundary.

Advantages of Using Dynaviews

An advantage of using dynaviews is that only the areas you want to see on a particular sheet are shown through a dynaview, and these areas are updated automatically in the dynaview as you make changes to your project file. You can create a different dynaview for each sheet, assign a different plot scale to each dynaview, create several dynaviews for a single sheet, or place a single dynaview on several sheets. Dynaviews also enable you to view each plot sheet before you actually plot it.

Dynaviews allow you to combine portions of plan and profiles together on a single plot sheet to plot survey sheets or construction plans, while you make all modifications in the original model. You can also enlarge portions of a project to show details, or show the same detail on multiple sheets.

Commands Associated with Dynaviews

You can use the commands in Table 4-8 to create and manipulate dynaviews:

Table 4-8 Dynaview Related Commands

Command	Menu Location	Operation
Pline	Draw/Pline	Creates a boundary for a dynaview.
Box	Draw/Pline	Creates a boundary for a dynaview.
PlotBox	Draw/Pline	Creates a boundary for a dynaview.
Dynaview	Draw	Creates a dynaview that displays the objects within a given boundary.
Reference	Modify	References objects to the dynaview boundary.
Pssheet	ASAP-planset / Sheet assembly	Automatically creates drawing sheets by assembling dynaviews and other objects within the sheet viewing mode.
DisplaySet	Settings/ Display Settings	Quick dynaview option - Displays the outline of the dynaview without showing the images of the objects inside of the boundary.

How to Select Dynaviews

Whenever you select a dynaview, position the pick box over the location of the dynaview boundary origin and click the left mouse button.

Understanding Attributes

A user definable attribute is a non-graphical type of object. The definitions for these types of objects are stored in user definable ASCII attribute definition files. User definable attributes are created with TML commands and must be referenced to a graphical object. User definable attributes are stored with the project file and are automatically deleted when an object is erased.

Attributes are used to store the properties of pipes and nodes and the results of the computations as attributes of the sets and nodes. Attributes are also used for basins and ponds. User definable attributes are a perfect way for the user to store additional data in a **Terramodel** project file.

User defined attributes are used and controlled by TML commands. For complete information about creating and using your own user-defined attributes see the TML On-Line help system.

Referencing Objects to Other Objects

Every object in **Terramodel** has a *record number*. The first object **Terramodel** creates has number 1, and the record numbers increase up to the current maximum number of objects established in the system configuration. Every object (with the exception of points) also has the ability to store as its property the record number of another object. This second property is called the *reference record number*.

The record number and the reference record number allow **Terramodel** to understand the relationship between two objects. Using the **Reference** command, you can assign or “reference” multiple objects to an object. In turn, these objects can be referenced by other objects, creating a branching system. By tracking this simple referencing system, **Terramodel** can determine all of the “ancestors” and “descendants” of any object and use this system to relate objects to the appropriate alignment.

In the process of surveying, designing and drafting (modeling) a project, there are some objects that only have a valid meaning when they are related to another object. For example, when the **Profile** command generates a profile along the center of a roadway, the new profile (a polyline) is only valid for that centerline. **Terramodel** stores the polyline in the profile view and references it to the roadway alignment. The coordinate system of the profile is typically displayed in stations and elevations, with the measurement of the stationing relative to the alignment.

But using the profile view's coordinate system presents some special considerations. For a single alignment, everything shown in the profile view should be related to the alignment. Many projects, however, have multiple alignments with overlapping coordinate systems. Additionally, other objects drawn with the profile of the alignment should be stationed with respect to the main alignment. (A typical case is when you want to plot the ground lines for the left and right rights-of-way.) And alignments are typically longer than what can fit on a single plot sheet. CAD-only systems force you to create copies of sections of the alignment for connecting plot sheets, which exposes you to errors when changes are not recorded in all copies.

By using object referencing and dynaviews, **Terramodel** solves the problems of combining multiple plan and profile views, updating multiple copies of the same area that show different aspects of a project, and displaying only the pieces of a project that are applicable to a specific task. This is accomplished with the concept of an active alignment.

Active Alignments

When you set an active alignment with the **Active** command, **Terramodel** uses it as the default in any commands that require you to select an alignment. **Terramodel** also intelligently references certain objects that you create to the active alignment. (For the **Profile** command, **Terramodel** automatically references the vertical profile it creates to the alignment.) Once an alignment is active, if you work in the profile view you will only see the objects that are referenced to it or that have no reference. This feature is very valuable for un-cluttering the screen and displaying only what is pertinent to the project area you are working on. To re-display all of the objects in the profile view, simply clear the active alignment.

Active alignments also allow **Terramodel** to station a secondary profile based on another alignment. If you have a primary alignment and would also like to create a profile of the ground along some other offset alignment using the stationing of the active alignment, the process is simple:

- Select the primary alignment as the active alignment.

- Use the **Reference** command to reference the offset alignment to the active alignment.
- Use the **Profile** command to profile the offset alignment. **Terramodel** properly interpolates the offset, and plots the results based on the stationing of the main (active) alignment.

This concept also works in the Sewer module. The **Pipedraw** command uses the stationing of the referenced alignment to draw the linework of the pipe branch. The text of the length of the pipes represents the distance from node (manhole) to node (manhole), but the lengths of the lines will reflect their position as stationed from the main alignment.

Chapter Five

Understanding Terramodel CAD

View Modes

Traditional CAD systems store all objects in a single 3D coordinate system. **Terramodel** enhances this functionality for traditional civil engineering applications by having different coordinate systems designed specifically for land modeling and the drafting of the model. There are eight *viewing modes* available to you while you work in your project. Five of these have a special purpose and three are available for future use. These view modes emulate the traditional plan view, profile view, cross section view, and the sheet view where plan, profile and cross sectional images can be placed within finished drawings.



The eight views operate independently of each other—i.e., if you change the horizontal alignment in the Plan view, the profile of that alignment is not dynamically affected. Most of the **Terramodel** commands operate in all of the views, although there may be some restrictions. When you try to use a command in an inappropriate view, your cursor will convert to the symbol shown in the left margin.

You see each of the views within windows which can be moved, sized, opened and closed. You can have multiple windows of the same viewing mode, or only use one type of viewing mode within a project. Some view commands, such as **Pan** and **Zoom**, can operate between views where the first data point or window corner can be specified in one view, and the second can be specified in another view of the same view mode. You can also copy or move objects between views, but only if the views are of the same viewing mode (plan, profile, etc.). The **ReView** command can be used to change the view mode of selected objects.

Plan View

The Plan view projects a “bird’s eye” view of your project while you work in the X and Y dimensions (east and north coordinates). This is the traditional coordinate system in which projects are designed and drafted. If your Coordinate Scroll is open, you can track the north and east coordinates, delta coordinates, bearing-distance, station and offset, layer-elevation, and slope-aspect readout in this view. All DTM data should be created in the plan view.

Profile View

The Profile view looks at your project from an elevation perspective using coordinates of station and elevation. You have the option of varying the vertical exaggeration with the *View Settings* dialog box (**SETTINGS Menu**, “View Settings”, the **ViewSet** command, in the On-Line Help system). While you are in this view, you can track within the Coordinate Scroll, the station and elevation coordinates, delta coordinates, grade-distance, and the station and delta elevation with regard to a vertical alignment. Use this view to store your vertical alignments and other profile and to look at surface profiles created by the **Profile** command.

Sheet View

The Sheet view lets you set up special drawing sheets that can contain dynaviews of all or portions of the other views. The sheet view functions in sheet based X and Y units (tracked in the Coordinate Scroll) at a scale relative to the Plan or Profile view. Here you can use the **Dynaview** command to establish plan, profile, cross section and combination plan/profile sheets for construction drawings.

Xsection View

The Xsection view is where the **RDX** and **XSHEET** commands create cross sections. The working units in this view, displayed and tracked in the Coordinates window, are Offset and Elevation. Dynaviews of the cross-section drawings are placed within the sheet view to construct cross-section sheets.

The cross-sections created in this view are both related to the active alignment and a specific station. The **ActiveStation** command is used to select the active alignment and active station. When in the cross section view, only objects referenced to the active alignment that have a station value that is within a unit (foot or meter) of the current active station or that have no elevation or are not referenced, will be displayed. When plines, text and blocks are created in the cross section view they are automatically assigned a reference to the current active alignment and given a Z axis value that is the current active station.

Superelevation View

The superelevation view is used to create and work with superelevation diagrams for roadways. The working units in this view, displayed and tracked in the Coordinates window, are Station and Slope.

Views 6–8

Three other view modes, numbered six through eight, are available for future use or other special purposes. You can establish their characteristics to suit your needs.

View mode properties

Each view mode has independent settings for controlling the view name, XYZ coordinate label, ordering of coordinates (XY or YX), the ability to apply plot scale factors when entering dimensions, the ability for dynaviews to exclude objects which are referenced to other dynaviews, the vertical exaggeration, grid settings and grid snap settings.

As an object is created it inherits a view mode as a property. Typically this is the active view mode although some commands are used in one view and create objects in another view. An example of this is when you use the **profile** command to create a profile along an alignment from a DTM layer. The alignment is selected in the plan view but the profile is created in the profile view.

Working with Layers

Layers are primarily used for grouping common objects together so they can be manipulated as a whole. Layers can also help in organizing a project file.

Each object in **Terramodel** is created with the property of a layer. Generally most users refer to an object as being created on an assigned layer. When you create an object, it is assigned the default object and point colors and linetype of the layer on which it is created. You can later modify an object's color and linetype after it is created.

Each layer can be used as a DTM surface. All 3D points on a layer are used as part of the DTM. The set lines that connect 3D points on the same layer are used as breaklines. These considerations should be used when deciding what objects to place on which layer.

The **LayerSet** command can be used to turn the visibility of layers on and off and to isolate a single layer, i.e. make it the only one visible. You may also toggle the ability to graphically snap to objects on selected layers. If there are groups of layers that should be viewed together, excluding all others, to represent specific drawing content, you may create a named layer list using the **LListSet** command. This layer list can then be used in the **LayerSet** command to select those layers and turn their visibility on. This layer list can also be made a property of a dynaview so that the dynaview only displays the included layers, and does so regardless of the current layer visibility within the target viewing mode. Layer lists are discussed in more detail in the following section.

Individual objects can also be turned *on* or *off* using the **On** or **Off** commands. This controls just that object's visibility, not that of the entire layer. As a general rule, most users will typically turn whole layers on and off, rarely toggling the visibility of individual objects.



Terramodel reserves layer 0 (zero) for special purposes. Points on layer 0 cannot be linked or labeled, and when you link other layers, duplicate points are transferred to layer 0 for checking. We recommend that you not use layer 0, leaving it reserved for the program's use.

Choosing Layer Names

Careful layer naming can make all **Terramodel** commands easier to use. For example, if you name one layer E_CONTOURS and another E_INDEX CONTOURS, they will appear together in an alphabetical list of layer names. You may also wish to use only characters that are supported by other popular CAD packages to improve compatibility for file translation.

Some types of selection options also let you use a layer mask for the names. When selecting with a mask the “*” character can be used as a wildcard to select names. We suggest that you assign layer names for similar groups of items using a common prefix or suffix. For example, utilities might be stored on the layers UTILWATER, UTILSEWER, UTILSTORM, UTILCATV, etc. These layers could then all be selected in a single operation using a click and drag motion of the mouse in the *Select by layer(s)* dialog box since they will be arranged together alphabetically. When editing the selection expression, a layer wildname of UTIL* can be used to select all layers matching the mask.

Assigning Layer Colors

Each layer has two colors associated with it. These represent the colors which are initially assigned to objects as they are created on that layer. An object’s color can thereafter be changed from the default color assigned at its creation, to any other color. The color of an object is a property of the object, not the layer on which is located. When you edit the default color(s) assigned to a layer, the colors of existing objects on that layer remain as they were. In other words the colors assigned to the objects themselves are not affected by the change in the layer’s default color assignment. There is, however, one exception where the color of an object can appear to be a property of the layer on which it is placed. When you assign color 0 to an object, it is always displayed as the default color assigned to the layer on which it is located. When you change the default color assigned to a layer, all objects located on that layer that are assigned color 0 change to display the revised color. When you reassign such an object to a different layer, its color may change to reflect a different default color as assigned to that layer.

As indicated above, a layer has two different default color assignments. The first, designated simply as the layer’s **Color** or object color, is the color assigned to all objects with the exception of points. The second, designated as the **Pt color**, is that assigned to point objects in particular. This allows points and sets to be two distinct colors when creating a set, potentially making it much easier for you to distinguish the points from the attached set. You can assign a layer’s default object and point colors with the **LayerSet** command. It is accessed by clicking the L Set button on the Toolbar or by typing **LayerSet** at the command bar. For more information on the **LayerSet** command, see the section entitled “Layer Settings Dialog Box” below.

Layer Settings Dialog Box

L Set

You can open the *Layer Settings* dialog box by clicking the L Set command button on the Toolbar, or by entering **LayerSet** at the command bar. This dialog box lets you create or delete a layer or layers, as well as establish the characteristics for a layer.

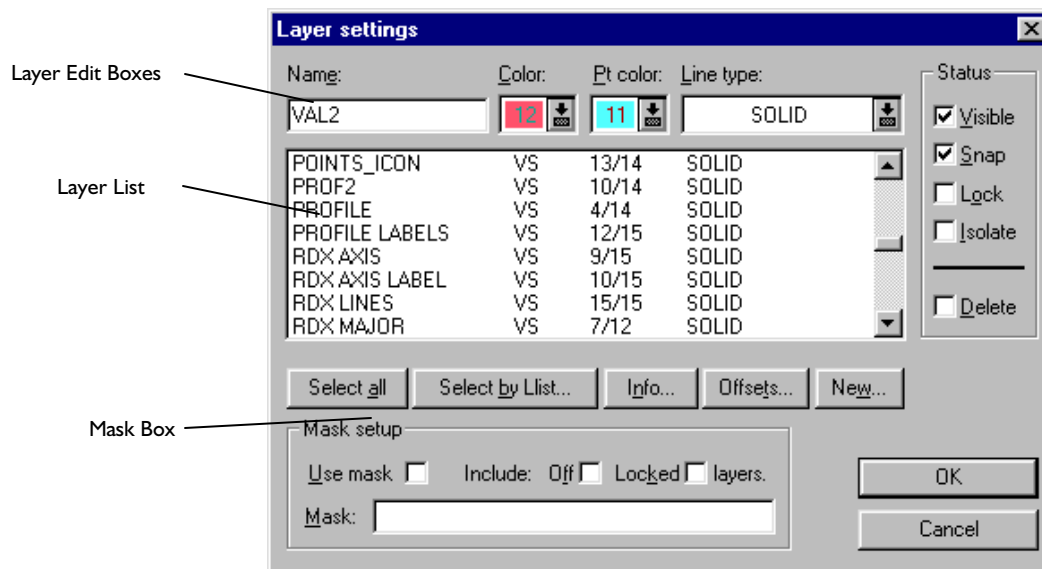


Figure 5-1 Layer Setup Dialog Box

The *Layer Setup* dialog box is used to control several properties in addition to creating and deleting layers. To select a single layer for editing, simply click the desired name in the layer list box. The layer will be highlighted in the layer list box and its name will be displayed in the Name edit control. To select multiple individual layers, press and hold CTRL while clicking your layer selections. To select a sequential range of layers, select the first, then press and hold the Space bar while clicking the last. To select all of the layers assigned to a particular layer list, use the Select by List button. To select all of the layers in the layer list box, use the Select All command button.

You can limit which layers will be listed in the layer list box by using the mask setup options described later.



Changes in color and linetype properties do not change the properties of any existing objects—only the default color and linetype of objects that will be created later.

Name

This edit text box is used to change the name of the selected layer. This control is only available when a single layer is selected.

Color

This color control is used to establish the default color to be assigned to all objects, except points, when initially created on the selected layer(s).

Pt color

This color control is used to establish the default color to be assigned to points when initially created on the selected layer(s).



Whenever an object has a color number of 0, the object's color is displayed by layer. Terramodel uses the current color setting of the object's layer to draw the object.

Linetype

This linetype control establishes the default linetype that will be used for lines created on the selected layer(s).

Status

These check boxes control the properties of layer display and selectability. None of the status properties are applied until the OK command button is selected. All status changes accumulate while you are in the *Layer Setup* dialog box until you click on OK. For instance, if you change the status of certain layers and then use the mask setup to display only a subset of those layers, all status changes are applied when the OK command button is selected—regardless of the layers currently displayed in the layer list box. **Terramodel** displays the current status of each layer in the second column of the layer list box.

- ☉ **Visible**—This check box controls the visibility of the selected layer(s). In addition to layer visibility, each object also has its own visibility status. While a layer is visible you can control the visibility of each individual object on that layer. When a layer's visibility status is off, objects will not be displayed regardless of their own visibility status. Objects that are not displayed cannot be selected graphically, but they can be selected by other direct means such as by color, layer, linetype etc. **Terramodel**

indicates that a layer is visible by displaying a “V” in the second column of the layer list box.

- ⊕ **Snap**—This check box controls whether you can select or snap to objects on the selected layer(s) with the cursor. When you can snap to objects on a layer, **Terramodel** displays an “S” in the second column of the layer list box.



*Snapping to objects requires time to search the database. You can perhaps reduce the amount of time needed and improve response by using the Running snaps selection on the SETTINGS menu, the Snap option of the **LayerSet** command, and the Search Settings to reduce the number of objects that must be considered.*

- ⊕ **Lock**—This check box lets you lock and unlock selected layers. When a layer is locked objects on that layer can not be edited or deleted. When a layer is locked objects can not be added to the layer.
- ⊕ **Isolate**—This check box lets you display only the objects on a selected layer in the graphics area. The visibility of all other layers is temporarily turned off. You can only isolate one layer at a time. When you turn off the isolate status, all other layers return to their previous display status. When a layer is isolated, **Terramodel** displays an “I” in the second column of the layer list box.



When a layer is isolated, you cannot turn off its visibility status.

- ⊕ **Delete**—This check box deletes selected layers. When the delete status is on, the V, S, L and I status indicators are replaced with “####” and the check boxes in the Status column are dimmed. **Terramodel** does not delete any selected layers until you click OK.



A layer must be empty, in all views, before it can be deleted.

Select All

This command button is used to select all of the layers in the layer list box. When you select all layers only the visibility, and snap status options can be set.

Select by List

This command button opens the *Layer list selection* dialog box where you can select or create a named list of layers. You can limit the layers shown in the layer list box to those which are members of a named layer list.

Layer Info

This command button opens the *Layer Info* dialog box. You can display the number of points on a selected layer as well as the minimum and maximum coordinate extents of the points on that layer. You cannot edit any information shown in this box.

Offsets

This command button opens the *Layer Offset* dialog box, where you can enter profile elevation and station offsets for the highlighted layer.

New

This command button opens the *New Layer* dialog box for creating a new layer.

Mask Setup

This section lets you reduce the number of layers shown in the layer list box to those layers having one or more common characters in their layer names. The abbreviated list keeps you from having to scroll through the entire list box to find specific layers.

- ☛ **Use mask**—This check box toggles the usage of a mask off and on.
- ☛ **Include off/locked layers**—These check boxes enable you to optionally include layers having their visibility status off and/or their lock status on.
- ☛ **Mask**—This edit control specifies a mask name. The mask supports all wild-card characters as defined in chapter 6, **Using Terramodel Custom Controls**. An example of a mask name is *CON*, where **Terramodel** lists layers with “CON” anywhere in the layer name.



Each time you change the mask phrase or include options, toggle off the Use Mask check box, then toggle it back on to update your layer list.

Layer Offset Dialog Box

Open the *Layer Offset* dialog box by clicking the *Offsets* button on the *Layer Setup* dialog box. Enter profile elevation and station offsets for the layer highlighted in the *Layer Setup* dialog box in these real number controls. This option is primarily used for compatibility with previous versions of **Terramodel**, it is not recommended that layer offsets be set to any value other than zero.

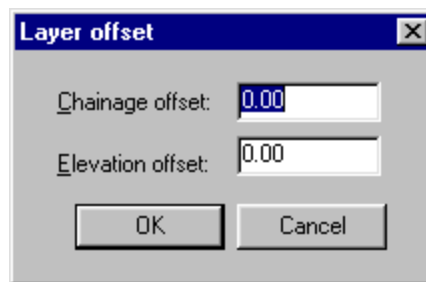


Figure 5-2 Layer Offset Dialog Box

New Layer Dialog Box

Open the *New Layer* dialog box by clicking the *New* button on the *Layer Setup* dialog box. To create a new layer, you define a unique name for the layer (limited to 16 characters), object and point colors, a linetype and the visibility, snap and lock status settings. The initial default settings for the new layer are the same as the currently-selected layer. For information on these controls, see under the *Layer Setup* dialog box, page 132, above.

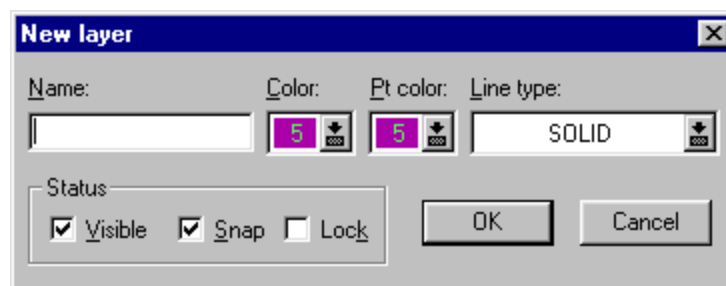


Figure 5-3 New Layer Dialog Box

Layer List Selection Dialog Box

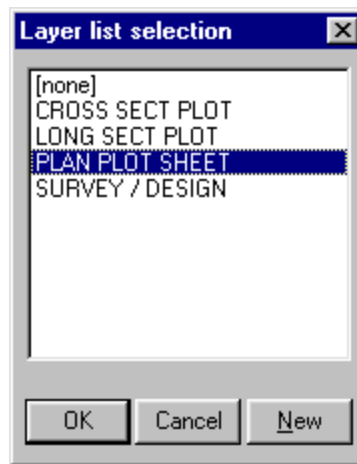


Figure 5-4 Layer List Selection Dialog Box

When you wish to display only those layers in the layer list box that are members of a particular layer list, click on the Select by list command button. This displays the *Layer list selection* dialog box as shown above, where you can select from any of the currently defined layer lists. For more information about layer lists, see the section entitled “Working with Layer lists” below.

Current Layers and the Layer Selection Control



All new objects created in **Terramodel** are stored on the “current” layer. You will change the current layer often during the course of a project editing session, as you organize your objects by layer. The layer selection control on the Toolbar displays the name of the current layer and lets you select a new current layer. Click the layer name in the control, then pick any object on the graphics screen and the current layer will change to the one on which the selected object resides. You can also use the *Layer Selection* dialog box (see below) to set the current layer, or add a new layer.

Layer Selection Dialog Box

Click the down arrow at the right end of the layer control on the Toolbar, or double click the layer name in that control, to open the *Layer Selection* dialog box. Select a current layer from a list of layers, or click New to create a new layer (see the *New Layer* dialog box, page 136).

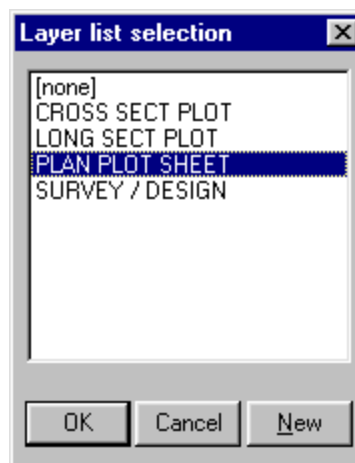


Figure 5-5 Layer Selection Dialog Box

Moving Objects to Other Layers

You may want to move objects from one layer to another. Use the **ReLayer** command to select the objects you want to move, and designate a different or a new layer on which to place them. To learn how to use the **ReLayer** command, see **MODIFY Menu**, "ReLayer", in the On-Line Help system.

Working with Layer Lists

Layer lists provide a mechanism wherein you can establish which specific layers are to be displayed within a dynaview. By assigning a layer list to the dynaview, a drawing composed of dynaviews can be made to always display the intended content, regardless of which layers are actually visible or invisible within the plan, profile or cross section views. You can also use a layer list to restore a particular combination of layer visibility or snap status settings.

Layer lists further provide a means by which *object prominence* can be varied as seen in different drawings. An object's prominence is an indication of its relative importance in the context of a particular drawing. Its prominence can be varied with raster plotters by using screened lines to reduce prominence, or by using finer line weights, or both. As an example, you may wish to create a drawing sheet in which sanitary sewer lines and their related callouts are the main objects of interest. You may wish that these main objects be drawn in the *foreground*, i.e., drawn as 100% solid objects perhaps using bold lineweights. You may wish that other subordinate objects, such as the existing topography, proposed water mains, storm sewers, etc., be drawn in the *background*, i.e., drawn as 20% or 50% screened objects, or with finer lineweights. The prominence of a given layer can be assigned as a property of the layer list which is associated with a dynaview. In defining the layer list to be assigned to the above drawing, certain layers can be designated as being in the foreground, while others are placed in the background. You can then create another layer list, perhaps containing the same set of layers, but in which the prominence assigned to certain layers is different. In the case of the above noted example, a second drawing sheet can contain another dynaview of the same plotbox, to which the second layer list is assigned. It can show the storm sewers in the foreground, while the sanitary sewers are moved to the background, along with the topo, and water mains. These two sheets might display some of the exact same data, but the colors of background objects, as displayed in the dynaview, can automatically vary from those actually assigned to the objects. Additionally, the pens used to plot these background objects can automatically be altered, producing the described adjustments in lineweight and screening as appropriate to the intended focus of each individual sheet.

Layer lists, therefore allow you to control both what is displayed in a drawing, and how it is displayed, without regard for the actual layer visibility and without needing to alter the object color assignments at the time at which you wish to create a plot. This exclusive **Terramodel** feature eliminates one of the most common headaches and complaints about traditional CAD systems — that of never being able to get the same drawing plotted twice with the same or proper content, and appearance. In order to get a sheet plotted as intended, you no longer have to turn layers on or off, or change object colors in order to screen certain objects. **Terramodel** takes care of this for you, without having to change any layer visibility or object color properties.

Layer List Settings Dialog Box

You can create a layer list and edit layer list settings using the *Layer List Settings* dialog box. It is opened by choosing **Layer list settings** from the **SETTINGS** menu.

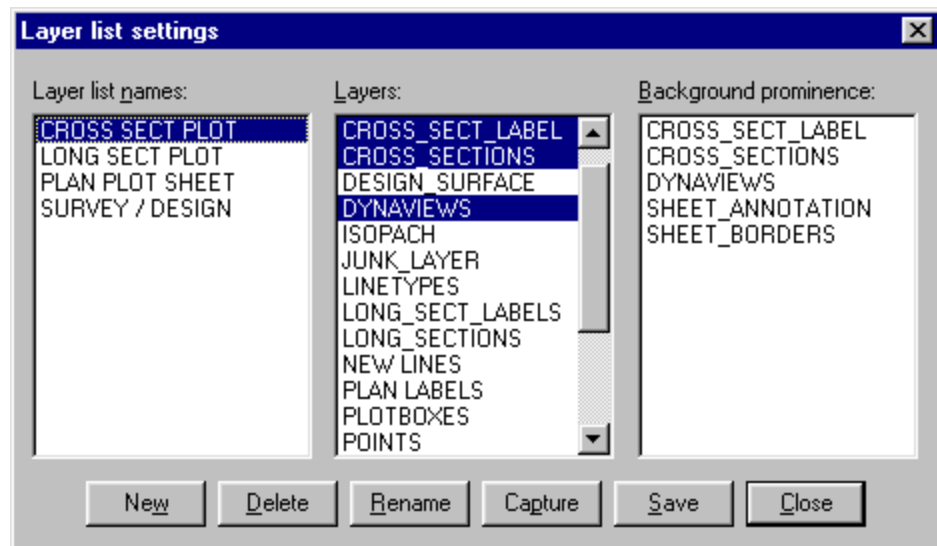


Figure 5-6 Layer List Settings Dialog Box

Layer List Names

This list box contains the names of each currently defined layer list.

Layers

This list box contains the names of each of the currently defined layers. The highlighted layers are those that are members of the layer list which is currently highlighted in the *Names* list box. As you select another layer list there, its current member layers are highlighted. To add a layer to the selected layer list, click on the layer name in the *Layers* list box to highlight it. To remove a layer from the layer list, click on a highlighted layer name to deselect it.

Background Prominence

This list box contains the name of each member layer of the selected layer list. Each layer is assigned foreground prominence by default, and foreground layers are not highlighted in this list. To display the objects on a given layer with background prominence, select the layer name in this list to highlight it.

New

This command button opens the *New Layer List* dialog box for creating a new layer list. See “New Layer List Dialog Box” below.

Delete

This command button allows you to delete the currently highlighted layer list.

Rename

This command button opens the *Rename layer list* dialog box, which allows you to rename the layer list.

Save

This command button saves changes to the layers assigned to the currently selected layer list. After clicking in the *Layers* list box to change the selected layers, you must click on the Save button to save the changes. If you select a different layer list after making changes to the current layer list settings, you will be asked if you wish to save those settings before selecting and displaying the other layer list settings.

Capture

This command button allows you to select the member layers of the selected layer list based on the current layer visibility settings. When you click on this button, all layers, which are currently visible, are selected. Layers whose visibility status is turned off are deselected. This lets you capture the current layer visibility settings within a layer list, so you can restore those settings later. After altering the selected layers you must click on the Save button to save the changes.

New Layer List Dialog Box

Open the *New Layer List* dialog box by clicking the New button in the *Layer List Settings* dialog box. Enter the layer list name in the Name text control. A layer list name is limited to 16 characters.

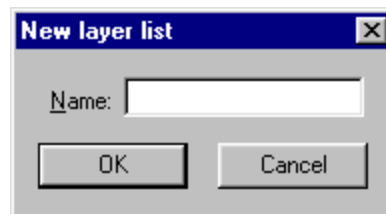


Figure 5-7 New Layer List Dialog Box

Rename Layer List Dialog Box

Open the *Rename Layer List* dialog box by clicking the Rename button in the *Layer List Settings* dialog box. Enter the new layer list name in the New Name text control. A layer list name is limited to 16 characters.

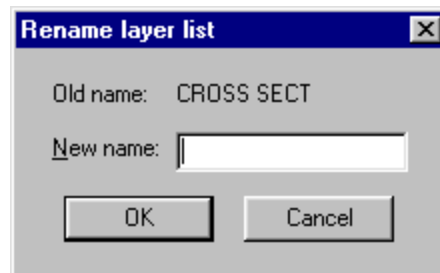


Figure 5-8 Rename Layer List Dialog Box

Assigning a Layer List to a Dynaview

You can control what is displayed within a dynaview by assigning it a layer list. When a dynaview is drawn it displays only those objects on layers specified within the layer list, regardless of the current layer visibility settings. When a dynaview is initially created it is not assigned a layer list (actually it is assigned a layer list named "none"). A dynaview that is not assigned a layer list always displays objects as they appear within the plotbox that it references. Assigning layer list "none" to a dynaview therefore does not mean that

it displays nothing. Instead, it means that it displays everything that is visible within the viewing mode into which it looks.

When you assign a layer list to a dynaview, it displays only those objects that reside on layers that are members of that layer list. It displays them even if the visibility attribute of that layer is turned off. The visibility of layers is set within the *Layer Settings* dialog box as described on page 133. Even though these objects will be invisible in the viewing mode into which the dynaview looks, they will be displayed in the dynaview. It will not display objects that reside on layers which are not included in the assigned layer list.

In addition to the visibility settings attributable to layers, the visibility of individual objects can be turned on or off. If an individual object is turned off, it will not display in a dynaview under any circumstances. For more information on turning an individual object on or off, see **MODIFY Menu**, “Object ON” and “Object OFF” in the On-Line Help system.

To assign a layer list to a dynaview, select **Layer List** from the **MODIFY menu**, or type **Llist** at the command bar. This opens the *Layer List Assignment* command bar where you can select a dynaview, and then select a layer list from a dialog box listing each of the currently defined layer lists.



Figure 5-9 Layer List Assignment Command Bar

For more information on selecting a layer list see **MODIFY Menu**, “Layer List” in the On-Line Help system.

Controlling Background Object Colors

When defining a layer list, you can designate that the objects on certain layers be displayed within a dynaview with background prominence as opposed to the default foreground prominence. When an object is to be displayed in the background, its color number can be shifted by a constant value which you can define. Therefore, for every color number intended as a foreground color, there can be an associated background color number which is derived by adding the designated color shift value. For more information on setting the background color shift, and enabling or disabling background color shifting, see **FILE Menu**, “Plot setup”, “Plot params...”, “Advanced” in the Terramodel online help.

When displaying background objects in a dynaview, the designated color shift affects only the on-screen display of the object’s color. The pen mapped to that shifted color is

not used in plotting that object. Instead, a pen shift is used, as described below, to plot background objects.

Controlling the Pen Selection for Background Objects

When the object displayed within a dynaview is on a layer designated to be shown in the background, its pen number can be shifted by a constant value, which you can specify. You might therefore want to define your plotter pens similar to that shown below, so that the shift will produce a screened line, or perhaps one with a finer lineweight.

Table 5-1 Example Pen Definitions

Pen #	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
Screened Percentage	100	100	100	100	100	50	50	50	50	50	20	20	20	20	20
Line Width	.25	.35	.50	.70	.90	.25	.35	.50	.70	.90	.25	.35	.50	.70	.90

Using the above example plotter pen definitions, a pen shift of 5 would create a background line of the same width as the foreground lines, but screened 50%. A pen shift of 10 would produce 20% screened background lines. For more information on setting the background pen shift, and enabling or disabling background color/pen shifting, see **FILE Menu**, “Plot setup”, “Plot params...”, “Advanced” in the Terramodel online help.

Working With Colors

Object colors are typically used primarily as a means of identifying and controlling the resulting plotted line characteristics, i.e. pen color, lineweight, and screened percentage in the case of raster plotting devices. This is accomplished by mapping colors to pens. Under certain circumstances, if you prefer, object colors can instead be used to differentiate among various feature types, disciplines, construction phases, etc. Colors can be used to make certain aspects of the project stand out on screen or fade into the background. Since plotter pen selection is normally based on color, most of the time you'll assign object colors based on the pen which you wish to use to plot the object. One exception to this, however, is text objects. You can instruct **Terramodel** to choose pens for plotting text, based on the plotted text height, independent of the object's color. For an explanation of automated pen selection based on text height, see "Text Plotting and Color Assignment" on page 167.

Color Selection Dialog Box



There are two color controls to the right of the layer selection control on the Toolbar. The color control on the left sets the color to be assigned by default to all object types created on the current layer, except points. The control on the right sets the color of points. Clicking the down arrow or double-clicking either color control opens the *Color Selection* dialog box. (See "CAD Controls" in chapter 6, **Using Terramodel Custom Controls** for more information about how these controls work.) With these controls you can assign to the current layer, new default colors for points and other objects. New objects created subsequently on that layer will be assigned these colors by default. By assigning one color to points and another color to the other object types, you can easily differentiate points from lines in a set.

The *Color Selection* dialog box displays a range of 64 color numbers at a time showing the corresponding colors. By clicking a range radio button you can view up to four ranges of color *numbers* (up to 255 color numbers), even though your computer may display fewer *colors*. Although objects can have color numbers from 0 to 255, **Terramodel** displays a maximum of 64 unique colors at a time in a repeating manner over the 255 color number range. Objects assigned a color of zero assume the default color assigned to the layer on which they reside. The 64-color maximum enables other applications to run within Windows without color conflicts.

Creating Your Own Palettes and Color Maps

Terramodel supports *color palettes* of up to 64 colors. The palette which is in effect defines all of the colors that are available. You can define as many different color palettes as your needs dictate, giving you control over the colors you see on the screen, within the limits of your hardware. The color palette definitions are stored in the [Palette] section of the TMODWIN.INI file. In defining a color palette, you assign each palette index number a color definition, i.e., you specify the red, green and blue values (0-255) that define the color. For more information on storing palette files, see chapter 15, **The TMODWIN.INI & P3SERVER.INI Files**, [Palettes] section.

The palette index numbers described above are not the color numbers that you see in the color controls when working with **Terramodel**. Instead, you work with color index numbers (0-255). Each of these color numbers are assigned a palette index number (a color definition). This is done using a *color map*. You can define multiple color maps to suit your needs. The colors you see within the *Color Selection* dialog box, as described above, depend on which color map is currently being used, and which palette it is associated with. You can select the current palette and color map using the **Palette + Colormap** selection on the **SETTINGS** menu, or the **Palette** command.

Mapping Colors to Pens

For the most part, the plotter pen used to plot an object is selected based on the color of that object. This is accomplished by mapping colors to plotter pens. Each of Terramodel's 255 colors are assigned to a particular pen number. An exception to this rule is found in plotting text objects when you've instructed **Terramodel** to select the pen based on the text height instead of its color. For an explanation of automated pen selection based on text height, see "Text Plotting and Color Assignment" on page 167. Another exception is when plotting objects with background prominence in a dynaview. In that case, the pen selection is based on a designated pen number shift from that pen number which would be used to plot that object in the foreground, based on its assigned object color. See the above section entitled *Working with Layer Lists* for an explanation of plotting with background prominence and pen number shifts. You can create as many pen maps as your needs dictate. These are stored in the [Penmaps] section of the TMODWIN.INI file. For an explanation of how to define and store pen maps, see **FILE Menu**, "Plot Params", "Plot Setup/Plot Params" in the On-Line Help system.

Working with Linetypes

Assigning different linetypes to layers and specific lines is another way to make line objects recognizable and to create a professional-looking, legible set of plans.

Loading a Linetype File

The **Terramodel** prototype project includes several basic linetypes. The optional “*add-ins*” files also include many more linetypes for special purposes. You can also create your own linetype files with custom linetypes.

To load a linetype file:

- Enter **LineTypeSet** from the **Terramodel** command line to start the command.
- Click the *Load* command button to load additional linetypes.
- Click the *Browse* button to select the desired linetype file.
- Select the linetypes from those listed to be loaded.

Once the linetypes are loaded, **Terramodel** will list them in the *Linetype* dialog box. The **LineTypeSet** command can also be used to purge selected lintypes. A project file is limited to 254 linetypes.

Changing the Linetype of the Current Layer



Use the Toolbar’s Linetype selection control to change the linetype of the current layer. If you click the linetype name in the control, you can pick any object on the graphics screen and the current layer linetype will change to the linetype of the object you selected. If you click the down arrow at the right end of the control, you will open the *Linetype* dialog box, where you can select a current layer linetype from a list of linetypes.

Linetype Dialog Box

Open the *Linetype* dialog box by clicking the down arrow to the right of the Linetype control on the **Terramodel** Toolbar (or the New Linetype control on the **Linetype** command bar). Select a current layer linetype from a list of linetypes.

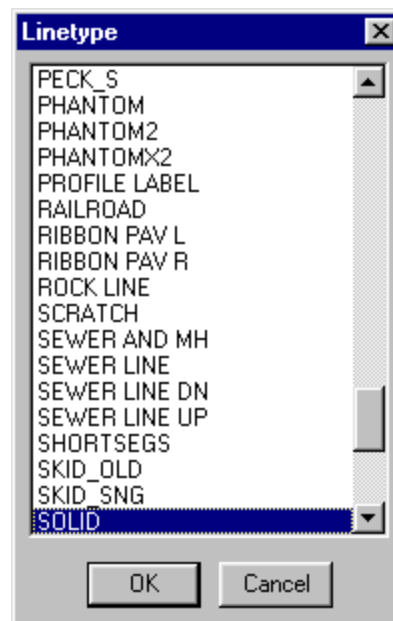


Figure 5-10 Linetype Dialog Box

Changing the Linetype of Selected Objects

To change the linetypes of objects that you select, use the **Linetype** command, or select **Linetype** from the **MODIFY** menu. Clicking the New Linetype control on the **Linetype** command bar also opens the *Linetype* dialog box (shown above).

Working with Linetypes

As a part of linetype definitions, you can include symbols in linetype patterns. **Terramodel** calculates and displays the symbol sizes and linetype repeat lengths for any linetype based on the current drawing scale (the sizes and lengths in the linetype definition are in actual plotter units). If you revise the drawing scale, **Terramodel** adjusts

the line pattern and symbol sizes and updates them with the next screen redraw. For this reason, if you want similar linetypes with different symbol sizes, you must create separate linetypes.

Redrawing the screen when you have linetypes with symbols can be very time-consuming, especially if the linetypes are complex. Whenever a line passes partially through the screen display, **Terramodel** must compute the entire line to have the pattern repeated in the proper position. To speed up line drawing, turn on **Quick Linetype** in the *Display Settings* dialog box and **Terramodel** will draw all linetypes as solid lines.

When you select lines with special linetypes, click along the center of the line as though it were a solid line. **Terramodel** does not check if a linetype symbol falls within a pick box because it would slow down the selection of lines.

Creating Custom Linetypes

Terramodel includes an ASCII linetype file (TMODEL.LIN) that contains several linetype selections that you can load into your project files. You can also create new linetype files (many users store linetype files according to line category), with each file containing a maximum of 255 linetype definitions.

Once you load a linetype file into a project, the associated .LIN file is no longer needed. If you load linetypes that already existed in your project file, they will be re-defined.

Each linetype definition consists of “pen up” and “pen down” combinations, where pen up movements are indicated by positive numbers and pen down movements are negative numbers. Linetype definitions can also include symbols from the SYMBOL.FNT font file described later in this chapter.

The Four Line Types

Of four basic types of lines, there are two groups that work in a similar fashion. Line types 0 and 1 have the ability to use a unique symbol at the beginning and end of a *line segment* and are *line segment* based. Line types 2 and 3 are based on the entire length of the *line*, and have the ability to place a unique symbol at the beginning and end of the *line*.

Line Types 0 and 1

Line types 0 and 1 calculate their line patterns based on the segments between points along a line. Line type 0 repeats the line pattern at a fixed length. For each new point on

a line, the pattern will begin again. This can result in a partial pattern at the end of each line segment.

Line type 1 varies the length of the pattern to include a whole number of patterns between any two points along the line. For the purposes of pattern repeating, points on lines can be point objects such as those contained in sets, or they can be calculated points such as the PC, PT, TS, ST, PVC and PVT, which are contained in polylines with complex curves. This line type can also include a parameter to indicate the maximum number of patterns between points.

Each of these two line types draws the beginning symbol at every point on the line except the last point, which receives the ending symbol. This lets you create a linetype that might, for example, be used for drawing sewer lines with manholes at each point. By having an initial symbol and an ending symbol (or no symbol at the end), **Terramodel** can draw multiple lines to the same point without creating conflicting symbols or duplicating the same symbol at the endpoint.

Line Types 2 and 3

Line types 2 and 3 calculate their line patterns based on the entire length of the line. Line type 2 repeats the pattern at a fixed length, does not begin again at each point on the line, and can have a single partial pattern at the end of the line. **Terramodel** uses this type of line for contour lines where the pattern should be smooth through each splined polyline.

Line type 3 adjusts the repeat length to have a whole number of patterns based on the entire length of the line. **Terramodel** can plot an initial and/or ending symbol for both of these line types.

The .LIN File Format

The first entry in the file designates the units of the file. Enter *%in*, *%m*, *%cm* or *%mm* to set inches, meters, centimeters or millimeters as the units of the file. The pattern will automatically be scaled based on the current measurement units to draw the linetype at the specified size.

Each linetype definition is stored in two lines of the .LIN file. A typical linetype can be defined as follows:

```
0, TEST 0, \040,0.10,-0,-0.05,\070\,0.10,-0,-0.05
\037,0.1,-0,0.4,-.1,\035,0.1,-0,-0.1,.4
```

The first line includes the following (each item is separated by a comma):

- Line type number (0 to 3).
- Name of linetype (maximum of 15 characters).
- A beginning symbol number (taken from the symbols shown in the symbol chart on page 171). The symbol number is preceded by a backslash and should always contain three digits.
- Beginning symbol size in sheet units defined by the **MeasUnit** command.
- Offset of beginning symbol from the centerline (a negative value places the symbol to the left of the center of the line, a positive value places the symbol to the right of the line center, 0 places it on the line)
- An initial pen movement before the pattern begins (this can start the line pattern a short distance away from the endpoint to allow room for a beginning symbol).
- An ending symbol number.
- Ending symbol size in sheet units defined by the **MeasUnit** command.
- Offset of ending symbol from the centerline.
- Distance from the end of the last pattern to the end of the line (to allow room for placement of an ending symbol).

Line type 1 can optionally include a parameter that specifies the maximum number of patterns between points. This parameter is included after the linetype name and before the beginning symbol number. Typically, it is not specified.

The second line defines the line pattern. A negative number indicates a pen up movement while a positive number indicates a pen down movement. The example above contains two different symbols in the line pattern. A description of the contents of a line pattern with symbols is discussed below.

Using Symbols in Linetypes

A symbol is described in a linetype definition in three parts:

- The symbol number, beginning with a backslash (\) character.
- The symbol size in sheet units defined by the **MeasUnit** command.
- The offset of the symbol from the centerline. A negative offset places the symbol to the left of the line while a positive value places the symbol to the right.



If a linetype definition contains only one entry, Terramodel assumes an error and draws the line as a solid line.

Terramodel draws all symbols center- and middle-justified. Because of this, we have adjusted some of the symbols in the TMODEL.LIN file so that they will be drawn properly. For example, two arrowhead symbols (symbol numbers 199 and 200) were created so that their center points are at the tip of the arrowheads, making them suitable for drawing leader arrows and dimension lines.

Terramodel draws all symbols based on the direction in which the line was created. Consequently, some symbols may appear upside down based on the direction of the line. To determine how a specific symbol will be drawn:

- Create a new text style using SYMBOL as the font.
- Set the style justification to center and middle.
- Place a point in your project file.
- Using the **Text** command with Point snap on, click the point you placed and enter a backslash (\) and the symbol number for the text.

Example Linetype File

Figure 5-11 shows the TMODEL.LIN linetype file. Use this file as a guideline for the format of your linetype file as well as the structure of your linetype definitions. The TMODEL.LIN included may vary from that shown below:

Figure 5-11 TMODEL.LIN Linetype File

```
# THIS FILE CONTAINS STANDARD SYSTEM LINE TYPES
#(the sizes are in inches)
%in
# Line type DOTS EVERY .2 UNITS
2,DOTS
0.01,-0.19
#
# Line type DASH EVERY .5 UNITS
2,DASH
0.5,-0.5
```



```
#
# Line type LONG DASH
2, LONG DASH
0.75, -0.25
#
# Line type LONG DASH DOT
2, LONG DASH DOT
0.8, -0.089, 0.002, -0.089
#
# Line type LONG DASH SHORT DASH
2, LONG SHORT DASH
0.7, -0.1, 0.1, -0.1
#
# Line type LONG DASH TWO SHORT
2, LONG TWO SHORT
0.5, -0.1, 0.1, -0.1, 0.1, -0.1
#
# Line type CONTINUOUS
0, shortsegs
.1, .1
#
# Line type DIMENSION LINE
2, DIMENSION LINE, \200, 0.20, 0.0, -.0, \199, 0.20, 0.0, -0.0
100, 100
#
# Line type LEADER LINE
2, LEADER LINE, \200, 0.20, 0.0, -0
1, 1
#
# Line type HATCH OUT (CLOCKWISE)
3, HATCH OUT
0.3, \37, 0.05, -0.0250, .0001
#
# Line type DEPRESSION (SAME AS HATCH IN)
3, DEPRESSION
.3, \37, .05, .0250, .0001
#
# Line type CENTER LINE (1.5 CENTERS)
3, CENTER LINE
```

```
0.65,-.1,\208,0.1,0.0,-0.1,0.65
#
# Line type ELECTRIC LINE (1.5 CENTERS)
3,ELECTRIC LINE
0.65,-.1,\169,0.1,0.0,-0.1,0.65
#
# Line type FENCE LINE WITH AN X (1.0 CENTERS)
3,FENCE (X) LINE
0.4,-.1,\35,0.1,0.0,-0.1,0.4
#
# LINE TYPE FOREST LINE
3,FOREST LINE
-.002,-.125,\210,.25,0.0,-.125,-.002
#
# LINE TYPE FOREST OUT
3,FOREST OUT
.0001,-.125,\210,.25,.0,-.125
#
# Line type GAS LINE (1.5 CENTERS)
3,GAS LINE
0.65,-.1,\171,0.1,0.0,-0.1,0.65
#
# Line type RAILROAD
3,RAILROAD
-.0001,\222,.2,.0,-.2001
#
# Line type rocks
3,ROCK LINE
-.0001,\213,.1,.0,-.15
#
# Line type sewer line
3,SEWER LINE
0.65,-.1,\183,0.1,.0,-.1,.65
#
# Line type SEWER LINE WITH MANHOLES
1,SEWER AND MH,\040,0.10,-.0,-.05,\040,0.10,-.0,-.05
0.65,-.1,\183,0.1,-.0,-.1,.65
#
# Line type stream
```

```
2,STREAM LINE
-.0001,.40,-.1,.01,-.1,.01,-.1,.01,-.2
#
# Line type swamp 1
3,SWAMP LINE 1
0.35,-.15,\211,0.2,0.0,-.15,.35
#
# Line type swamp 2
3,SWAMP LINE 2
0.35,-.15,\221,0.2,0.0,-.15,.35
#
# Line type telephone line
3,TELEPHONE LINE
0.65,-.1,\184,0.1,0.0,-.1,.65
#
# Line type WATER LINE (1.5 CENTERS)
3,WATER LINE
0.65,-.1,\187,0.1,0.0,-.1,0.65
#
# Line type 10" GAS LINE (1.5 CENTERS)
3,10" GAS LINE
0.65,-.1,\149,0.1,0.0,-0.11,\148,0.1,0.0,-0.11,\134,0.1,0.0,-
0.1,-0.1,\171,0.1,0.0,-0.11,\165,0.1,0.0,-0.11,\183,0.1,0.0,-
0.1,-0.1,\176,0.1,0.0,-0.11,\173,0.1,0.0,-0.11,\178,0.1,0.0,-
0.11,\169,0.1,0.0,-0.11,0.65
#
# Line type for profile labeling
1,PROFILE LABEL,\041,0.05,0.0,0.0,\000,0.0,-.0,-.0
100,1
```

Working with Text and Text Styles

Text Fonts

There are three basic supplied fonts that you can use to create text objects and attribute text in **Terramodel** project files:

- **TMODEL** is a fixed-spaced font usually reserved for tables and other documents where the text must remain column aligned.
- **TMODEL** is a proportionally-spaced font of the same typeface as above.
- **SYMBOL** is a font containing various symbols which can be mapped onto points.

For each of the above fonts, the original ASCII shape (.SHP) files, the compiled **Terramodel** font (.FNT) files, and the equivalent compiled AutoCAD (.SHX) files are furnished. The latter can be used when translating **Terramodel** files to AutoCAD and should be supplied along with the AutoCAD .DWG or .DXF file when you give someone a translated drawing. If you installed the optional “*add-ins*” and Acad font files when installing **Terramodel** you’ll have access to numerous other fancier fonts in addition to those mentioned above.

Font Shape files

For each of the supplied font files (.FNT) there is a font shape file (.SHP) supplied. Shape files are ASCII text files that describe the character number and the pen movements required to draw the character. **Terramodel** uses the same basic rules as AutoCAD for shape definitions, except that big-fonts and unicode font characters above 255 are not supported. Therefore some font shape files designed for AutoCAD can be compiled as **Terramodel** font files. The complexity of creating your own font from scratch places this activity beyond that which most users would normally attempt. However, anyone wishing to do so may use any good text book describing the creation of AutoCAD .SHP files as a guide.

AutoCAD users have created many public domain shape files that you can use to enhance **Terramodel**’s abilities. Search the internet for AutoCAD font files if you would like to use more fonts than are supplied. You should download the .SHP file and compile the file as described below.

Creating Fonts from Shapes

Shape files must be compiled into font files (.FNT) before they can be used by **Terramodel**. First you must bring up a DOS prompt window. From the DOS prompt go to the folder where you installed **Terramodel**. The FONTC program located there can be used to create the font file. Enter:

```
FONTC filename
```

Where *filename* is the name of the .SHP file without the filename extension. The .SHP file must therefore be located in that same folder. A font file (.FNT) with the same name will be created and stored in that folder.

Superscripts and Subscripts

Terramodel utilizes the same rules for superscripts and subscripts in font files as AutoCAD. Only the TMODEL.FNT file has superscripts and subscripts defined. The following steps describe how to add superscripts and subscripts to other shape files.

1. Edit the shape file with an ASCII text editor.
2. Replace the 221, 222, 227 and 228 character definitions with the following:

```
*221,8,sub_off  
2,004,2,8,(0,47),1,0  
*222,8,sub_on  
2,8,(0,-47),003,2,1,0  
*227,8,super_off  
2,004,2,8,(0,-47),1,0  
*228,8,super_on  
2,8,(0,47),003,2,1,0
```

3. Use Fontc to recompile the file.

In the above example the 47 and -47 values define the Y axis offset for the imaginary pen movement. These values may need to be adjusted for each font. The 003 code causes all following vector lengths to be divided by the integer following the 003 code. The 004 code causes all following vector lengths to be multiplied by the integer following the 004 code. In the above example the superscripts and subscripts are 1/2 the size of the unadjusted font. With these modifications, whenever **Terramodel** encounters one of the above characters in a text object, superscripts and subscripts are displayed.

Any specific ASCII code can be entered by entering the “\” character and the code number. To begin superscript enter \228 and the desired text. To end the superscript enter \227.

Creating Shape Files From SHX Files

Occasionally, you may find in the public domain a compiled (.SHX) font file that you would like to use. We have provided an .SHX decompiler that will decompile many, but not all, .SHX file types. Your use of the SHX2SHP decompiler is limited to the restrictions described in the Notice of Restricted Use as shown below. Read this notice *before* using this product.

To use the SHX2SHP decompiler, from a DOS box navigate to the folder where you have **Terramodel** installed.

From the DOS prompt enter:

```
SHX2SHP filename
```

Where *filename* is the name of the .SHX file. A .SHP file will be created. You may now use the font compiler as described above to create a **Terramodel** font.

Notice of Restricted Use

IMPORTANT NOTICE TO ALL USERS OF THE SHX2SHP DECOMPILER: The SHX2SHP program permits decompilation of certain font files available in the marketplace. Owners of such font files may claim copyright protection under the laws of the United States and foreign countries. It may be illegal to copy or prepare derivative works based upon such material for which you or your company do not own the copyright unless you have the permission of the copyright owner or unless your activity comes within some of the limited exceptions of the copyright laws. Copyright infringement carries with it serious civil and criminal penalties under the law. Do NOT

use the SHX2SHP program to illegally copy or prepare derivative works from copyrighted material.

Many owners also seek to impose restrictions on the use of their font files through license agreements. You should review any such license agreement to determine whether such restrictions apply to you.

If you have questions regarding legal uses of this product or copyrighted material, consult your attorney or the owner to the copyrighted material in question.

Text Leader Lines, Text Borders and Callouts

In addition to the text itself, a text object can include a *leader line* and/or a *border line*. A leader line is used to point to an object that is being labeled. The text object, the leader line, and perhaps a text border as well, can combine to form what is referred to as a *callout*. A callout usually will employ a leader line, but it need not. A callout is therefore any text object that specifically labels or displays information about another object or project feature. The object being labeled, and to which the leader line points, is called the *subject* of the callout. Example subjects might include sanitary manholes, curb inlets, curb lines, right-of-way lines, fire hydrants, or virtually any object or site feature in your project. The record number of the subject is an assigned property of any text object that functions as a callout. A callout often labels or displays certain attributes or properties of the subject. As an example, its coordinates, or its station and offset with respect to some alignment. In a case where the station and offset of the subject with respect to an alignment is to be labeled, the record number of that alignment must also be assigned as a property of the text object, so that it can display the computed station and offset of the subject with respect to that alignment. This is referred to as the callout's *subject HAL*.

The following section covers in detail, the mechanism by which text objects can be made to display attributes and computed properties of other objects. See “Understanding Text” in chapter 4, **Understanding Terramodel Objects** for more information about the supported leader lines and text borders. See **Edit Menu**, “Edit Object/Text” in the On-Line Help system for more information on assigning an object as the subject of a callout, or specifying the alignment associated with that subject.

Mirrored Text

Define the aspect setting of a text object as a negative value to cause the text to be drawn in a mirrored fashion.

Smart Text—Embedded Attribute Text (EAT) Codes

Smart text is an important, powerful and unique feature of **Terramodel**'s support for text and labeling. Using this feature, you can create text objects composed of phrases that contain textual references to certain project information, or to certain attributes or computed properties of the text object itself, or of other designated objects. As an example, you can create a text object as a callout that displays phrases which contain the name, northing, easting and elevation of a designated point, and its station and offset with respect to a particular alignment. What makes this text smart is that it doesn't actually contain the text displaying those attributes of that point. Instead, it contains a reference to the point and its attributes, so if the point is moved, renamed, or its elevation changed, the text automatically changes to reflect the current state of the data associated with that point.

You can create smart text by inserting an *Embedded Attribute Text* code (EAT code) within the text string, or using one as the entirety of the text string. EAT codes have been designed to reference several different kinds of information. In addition to the smart text that you can create, there are other forms of smart text that **Terramodel** creates, containing EAT codes that are not generally meant for use by the user. You'll still want to be aware of what these are so you'll recognize them when you see them. Table 5-2 contains a list of the EAT codes that you might typically use to create smart text. Table 5-3 lists the variables that can be placed within an EAT code to reference a particular attribute of an object. Table 5-4 lists the EAT codes that **Terramodel** uses internally. You'll not normally create text yourself with the EAT codes in this last table.

When editing a text object with the **Edit** command, the EAT code is displayed in the edit text control, not the resulting string. Therefore if you edit smart text created by a **Terramodel** command or a TML program, you will see some of the EAT codes contained in Table 5-2 or Table 5-4. You will not normally edit those EAT codes, which you did not create yourself, but you might often create and edit you own EAT codes. When a text object contains an EAT code that references another object, i.e., one of the \PAR or \SUB codes defined in Table 5-2, if the object it references is ever deleted, the smart text can no longer be displayed. In that case the EAT code itself will be displayed within the text string in most instances. You can then either delete the text object, or reference it to another suitable parent object or subject. In some cases, text that references an object's properties with an EAT code will automatically be deleted if the referenced object is deleted.

In order to make it easier for you to use EAT codes, those commands that create or edit text contain an EAT codes or EAT command button. This button opens the *EAT Code Insertion Aid* dialog box. It provides a simple means of placing an EAT code within a text string, without any knowledge of the EAT code syntax. So you'll normally use this aid to place EAT codes. When you use this aid, the EAT code is inserted at the text insertion marker, or it replaces the highlighted text within the underlying dialog box. The following tables are provided so that you can understand the EAT code syntax, though it

is not normally required that you understand it. However, though the EAT code insertion aid provides a number of date and time formatting options, if you desire a format other than those provided, you can accomplish that by manually entering the EAT code. See **Draw Menu**, “Text/Single line Text/EAT Code Insertion Aid Dialog Box” in the On-Line Help system for more information on this feature.

Table 5-2 EAT Codes

Code	Format	Function
OBJ	\OBJ{variable, [F/W,] [# of decimals]} (1), (2), (3), (4), (5)	Displays attributes of the text object itself.
PAR	\PAR{variable, [F/W,] [# of decimals]} (1), (2), (3), (4), (5)	Displays attributes of the text object's parent object, i.e. that to which the text object is referenced.
SUB	\SUB{variable, [F/W,] [# of decimals]} (1), (2), (3), (4), (5)	Displays attributes of a callout's subject object.
HAL	\HAL{variable, [F/W,] [# of decimals]} (1), (2), (3), (4), (5)	Displays the station and/or offset of either the callout text, its subject, or its leader line point with respect to the subject HAL.
PROJ	\PROJ{D[format codes]} (6) (date/time) \PROJ{P} (project file name) \PROJ{T} (project title) \PROJ{SHx, [# of decimals]} (plot scale) where x = viewing mode (0-7) \PROJ{SVx, [# of decimals]} (vert exagg) where x = viewing mode (0-7)	Displays general information about a project file.
PROJ	\PROJ{v,type,Project variable name} types: s = string, i = integer, d = double	Displays the content of the referenced project variable.
\n		A special code used within a line of text to start a new line. Allows a single text object to be displayed with multiple lines.

(1) Variable arguments are shown in Table 5-3.

(2) Parameters shown within brackets ([]) are optional.

- (3) The “F” parameter causes only the fractional part of real numbers to be displayed.
- (4) The “W” parameter causes only the whole number part of real numbers to be displayed.
- (5) Specification of the number of decimals applies only to real number attributes.
- (6) Optional date formatting codes are as follows:

MO	month
DA	day
Y	year
H	hour
MI	minute
S	second
DW	day of the week

The date formatting codes are not case sensitive.

You can use these date formatting codes along with literal text within the optional date format string to format the date/time display anyway you wish. The following examples show various date/time EAT codes and the resulting text as it will be displayed.

```
\PROJ{DMo Da, Y H:Mi:S Dw}   Jun 6, 1996 10:25:35 Thu
\nPROJ{DDw, Mo Da, Y}       Thu, Jun 6, 1996
\nPROJ{DDw H:Mi}           Thu 10:25
```

If you omit the optional format codes this EAT code will display as follows.

```
\PROJ{D}                   Thu Jun 6 10:25:35 1996
```

Table 5-3 EAT Variables

Variable	Represents
X	The X coordinate of either the text object (\OBJ), its parent object (\PAR), or its subject (\SUB).
Y	The Y coordinate of either the text object (\OBJ), its parent object (\PAR), or its subject (\SUB).
Z	The Z coordinate of either the text object (\OBJ), its parent object (\PAR), or its subject (\SUB).
P	The point number of either the text object's parent object (\PAR), or its subject (\SUB).
N	The name of either the text object's parent object (\PAR), subject HAL (\HAL) or its subject (\SUB).
L	The layer of either the text object (\OBJ), its parent object (\PAR), or its subject (\SUB).
R	The enclosed area of either the text object's parent object (\PAR), its subject (\SUB), or its subject HAL (\HAL), when such objects are either a pline or a set.
E	The perimeter measurement of the line object referred to as the parent object (\PAR), the subject (\SUB), or the subject HAL (\HAL). This line must be closed.
G	The total length of the line object referred to as the parent object (\PAR), the subject (\SUB), or the subject HAL (\HAL). The line may be either open or closed.
Sarg	The station of either the text object's insertion point, its subject, or its leader line point, with respect to the text object's assigned subject HAL. ⁽⁷⁾
Oarg	The offset of either the text object's insertion point, its subject, or its leader line point, with respect to the text object's assigned subject HAL. ⁽⁷⁾ , ⁽⁸⁾
A,args	The designated user definable attribute of either the text object's parent object (\PAR), or its subject (\SUB). ⁽⁹⁾

- (7) The argument which follows the “S” and “O” variable indicates the object for which the station or offset is to be reported. The supported arguments are as follows:
- O Reports the location of the text object’s insertion point.
 - S Reports the location of the text object’s subject.
 - L Reports the location of the end of the text object’s leader line.
 - N Reports the name of the HAL.
- (8) The offset variable can display several textual components in addition to the offset distance, This includes a representation of the offset direction, and can include literal text, a measure unit indicator, and punctuation. See **Settings Menu**, “Units Settings”, “Labels” in the On-Line Help system for more information on the settings that control offset labeling.
- (9) The required syntax for designating an attribute variable’s arguments is as follows:

A[,# of decimals]attribute name:field name[,element]

where *# of decimals* is optional, applying only to real number attributes,
attribute name is the name of the attribute,
field name is the name of the attribute field, and
element is the optional element number used for table attributes.

Table 5-4 Terramodel's Internal EAT Codes

Code	Format	Function
SEG	<p data-bbox="586 657 1040 684">\SEG=segment label style name</p> <p data-bbox="586 804 1040 989"> \DISTdc (straight segment length) \ARCLdc (Arc length) \MORDdc (Mid ordinate) \TANGdc (Tangent length) \CHRDdc (Chord length) \RADSdc (Radius of arc) </p> <p data-bbox="586 1062 1040 1209"> where d is the length suffix as presented in the <i>Label segments style settings</i> dialog box (0-3), and c is the flag indicates if dual units are to be used, if c=D then dual units are used </p> <p data-bbox="586 1251 1040 1308"> \BRGd (Straight segment bearing) \CHBRd (Chord bearing) </p> <p data-bbox="586 1377 1040 1465"> Where d designates the bearing format to be used (0-5), as defined in the <i>Label segment style settings</i> dialog box. </p> <p data-bbox="586 1507 1040 1556"> \DOC (Degree of curvature) DLTA (Delta of arc) </p>	<p data-bbox="1166 657 1398 783">Used by the LabelSeg command to create segment labels (Auto option)</p> <p data-bbox="1166 804 1398 926">Used by the LabelSeg command to create segment labels (User option)</p>

Text from External Files

A text object can also be used to display the contents of an external ASCII text file. To accomplish this create a text object as follows, using the **Text** command:

```
$File:filename
```

where *filename* is the name of the text file, with extension, for which you want to display the contents.

Terramodel will show the contents of the file using the properties of the text object (colors, layer, etc.), and let you select and manipulate the entire displayed contents as a single object. Once you have referenced an external text file in this manner, you can make changes to its contents and those changes will automatically be reflected in your project file. The spacing between the lines of text can be controlled by the *Space (%) between lines of text* integer control which is contained within the *Draft Settings* dialog box. See **Settings Menu**, “Draft Settings” in the On-Line Help system.

Drafting Text

Text Plotting and Color Assignment

Terramodel provides two different means of selecting the plotter pen when plotting text objects. At your option it will select a pen based on the color of the text object and the current pen map, or it can select a pen based on the text height. For the single stroked fonts that are commonly used in engineering drawings, the desired lineweight typically increases as the text height increases. Instead of requiring that you manually select a text color that will produce a lineweight appropriate for the intended text height, you can instruct **Terramodel** to always plot text at the appropriate lineweight (with the appropriate pen), regardless of its color. You can then use text color variations for differentiating text objects visually based on some consideration other than plotting.

You can designate six distinct ranges of text height, and assign a plotter pen to each range. Whenever the automated text height to pen mapping is in effect, **Terramodel** will determine within which range, if any, the text object’s height falls, and select the pen accordingly. If the text height falls outside of a defined text height range, the plotter pen is selected based on normal color mapping.

Automated text height to pen mapping is deactivated when pen sorting is turned on. See **File Menu**, “Plotter Setup”, “Plotter Configuration Dialog Box” in the On-Line Help system for an explanation of the Sort by Pen setting.

When automated text height to pen mapping is in effect and the text is on a layer which is to be plotted with background prominence, **Terramodel** will first determine the plotter pen based on the text height, and then shift it based on the designated background prominence pen shift. For more information on plotting objects in the background, see the section entitled “Working with Layer Lists” on page 139. See **FILE Menu**, “Plot setup”, “Plot params...”, “Advanced” in the On-Line Help system for more information on controlling the pen selection while plotting text.

Text Sizes

When text is plotted, the character size is measured from center of pen to center of pen. Standard Leroy template sizes, however, are measured from the top of the ink to the bottom of the ink. In order to match a particular Leroy size, you should plot the character to the desired size, less one full pen width. Table 5–5 below matches Leroy sizes to common line widths and letter sizes:

Table 5–5 Text Sizes

Leroy Template	Pen #	Line Width inches/mm	Letter Size inches
80	00	0.0120/.30	0.0680
100	00	0.0120/.30	0.0880
120	0	0.0140/.35	0.1060
140	1	0.0195/.50	0.1205
175	2	0.0235/.60	0.1515
200	3	0.0315/.80	0.1685
240	3	0.0315/.80	0.2085

For best results, you should not deviate more than one pen size from the suggested pen size shown for each template. You can set up these standards in your prototype file, so that they will already be stored in each new project file you create.

Text Orientation and Automatic Flipping

You can control the way a text object reacts when it is rotated, or when displayed in a dynaview that is rotated. You can also make a text string bend to follow a curvilinear alignment. This is done by setting the text object's Orientation.

When the orientation is set to *rigid*, as a text object is rotated, its insertion point is rotated about the designated rotation point, and its direction (displayed rotation angle) is altered by the rotation angle. As it is rotated, it is allowed to appear upside down, i.e. such that it does not read from the bottom or right. Rigid text that appears in a dynaview will be rotated as the dynaview is rotated and is allowed to read upside down in that instance as well.

When a text object's orientation is set to *fixed*, its direction is fixed and is not affected by rotation. As the text object is rotated, only its insertion point is rotated about the designated rotation point. Its direction is not altered. When appearing within a rotated dynaview, it will always be displayed in the same direction as it is shown in its native viewing mode, regardless of the dynaview's rotation. As an example, if all of the lot numbers in a subdivision are shown as horizontal text in the plan view, i.e., at a direction of S 90 00 00 E, and assigned a fixed orientation, then when displayed in the rotated plan view dynaviews of the typical plan and profile sheet, they will still be displayed horizontally. The text direction is not rotated as the dynaview is rotated, nor as the text objects themselves are rotated in their native viewing mode.

When set to *legible*, as a text object is rotated, either in its native viewing mode or via the rotation of a dynaview in which it is seen, its direction is changed as the text is rotated, as in the case of *rigid* text. However, it is not allowed to be turned upside down. It flips so as to always remains legible, i.e., it can be read from the bottom or right side of the screen or the sheet. This allows the same text object to be plotted from multiple dynaviews that have different rotations, to be displayed at multiple directions as a result, and still be read properly from the bottom or right side of the sheet. When a text object's orientation is set to *legible*, **Terramodel** honors a *flip over angle* setting. This causes text to flip to read legibly, only after it is rotated more than the specified number of degrees beyond the vertical. This setting is controlled by the *Flip over angle* integer control which is contained within the *Draft Settings* dialog box. See **Settings Menu**, "Draft Settings" in the On-Line Help system.

The *along* text orientation allows you to specify that the text bend and curve as necessary to follow along an alignment. Text with this orientation will follow the course of the alignment record to which it is referenced. If you assign this orientation to a text object that has no parent object, it will function in the same manner as if it had a rigid orientation. Text which is assigned the *along* orientation can not be rotated. It always follows its parent alignment's course, regardless of its own rotation settings, and is drawn in the same direction as the alignment object it references. If you wish that a text object be drawn along an alignment in a direction opposite to that in which the alignment is drawn, it can be assigned the *along flipped* orientation.

Text Styles

Terramodel gives you the ability to control the font, height, aspect ratio, slant, orientation and justification of text and to assign these properties to *text styles*. These variable text metrics give you a multitude of choices in the appearance of text. The appearance of a text object when created is based on the *current text metrics*. You can edit these current text metrics individually from within any of the text creation commands. Such revisions remain in effect until changed once again. You can set all of the text metrics at once, to match a predefined set of standard values, by selecting a text style as the current text style within the *Text Style Settings* dialog box. The text style is therefore a means of storing a standard, commonly used, set of text metrics. Text is not created in accordance with the current text style, but in accordance with the current text metrics. Selecting a text style simply sets the current text metrics to a predefined state.

Text Style Settings Dialog Box

When you execute the **StyleSet** command, the *Text Style Settings* dialog box opens to allow you to create, delete, or edit text styles (see **SETTINGS Menu**, in the On-Line Help system).

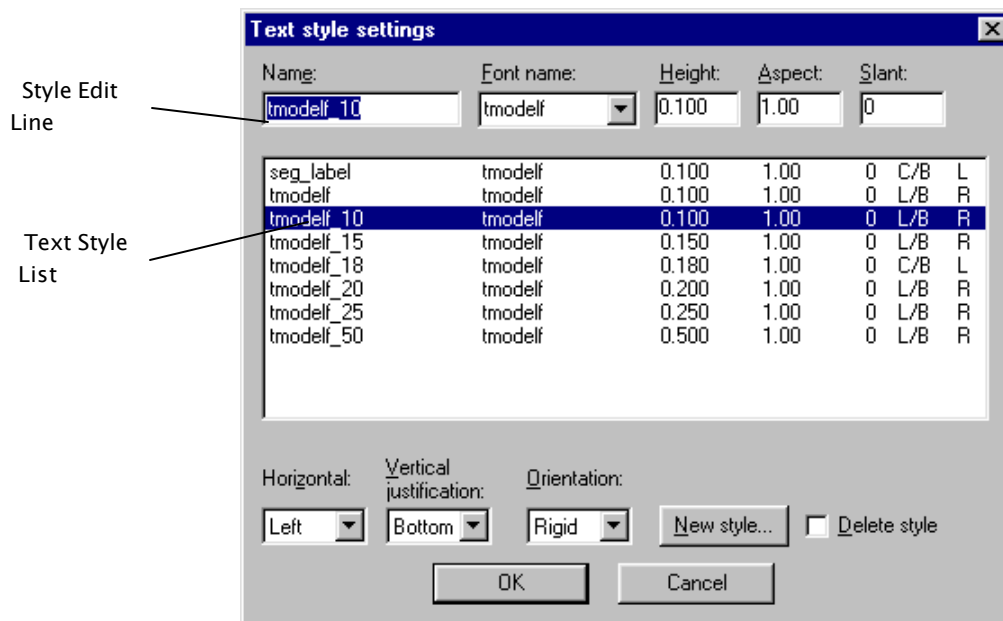


Figure 5-12 Text Style Settings Dialog Box

	0	1	2	3	4	5	6	7	8	9
10										
20										
30				·	+	×	-		◇	*
40	○	⊙	⊗	⊕	⊗	●	⊗	⊙	⊗	⊙
50	□	⊠	⊡	⊢	⊣	■	⊞	⊟	⊠	⊡
60	⊠	⊡	⊢	⊣	⊤	⊥	⊦	⊧	⊨	⊩
70	⬡	⬢	⬣	⬤	⬥	⬦	⬧	⬨	⬩	⬪
80	△	▲	▴	▾	◆	◇	▷	◁	↓	↑
90	⬠	⬡	⬢	⬣	⬤	⬥	⬦	⬧	⬨	⬩
100	⊛	⊜	⊝	⊞	⊟	⊠	⊡	⊢	⊣	⊤
110	⊥	⊦	⊧	⊨	⊩	⊪	⊫	⊬	⊭	⊮
120	⊯	⊰	⊱	⊲	⊳	⊴	⊵	⊶	⊷	⊸
130	⊹	⊺	⊻	!	"	#	\$	%	⊿	·
140)	(*	+	,	-	.	/	0	1
150	2	3	4	5	6	7	8	9	:	;
160	<	=	>	?	@	A	B	C	D	E
170	F	G	H	I	J	K	L	M	N	O
180	P	Q	R	S	T	U	V	W	X	Y
190	Z	Ⓢ	Ⓣ	Ⓤ	Ⓥ	Ⓦ	Ⓧ	Ⓨ	Ⓩ	▶
200	▶	◀	◁	▷	↖	↗	↘	↙	↕	↔
210	↗	↘	↙	↕	↔	↖	↗	↘	↙	↕
220	√	⊖	⊗	⊘	⊙	⊚	⊛	⊜	⊝	⊞
230	⊟	⊠	⊡	⊢	⊣	⊤	⊥	⊦	⊧	⊨
240	⊩	⊪	⊫	⊬	⊭	⊮	⊯	⊰	⊱	⊲
250	⊳	⊴	⊵	⊶	⊷	⊸	⊹	⊺	⊻	⊼

Figure 5-13 Symbol.fnt file

	0	1	2	3	4	5	6	7	8	9
10
20
30	.	!	"	#	\$	%	&	'		
40	()	*	+	.	-	.	/	0	1
50	2	3	4	5	6	7	8	9	:	:
60	<	=	>	?	@	A	B	C	D	E
70	F	G	H	I	J	K	L	M	N	O
80	P	Q	R	S	T	U	V	W	X	Y
90	Z	[\]	^	_	`	a	b	c
100	d	e	f	g	h	i	j	k	l	m
110	n	o	p	q	r	s	t	u	v	w
120	x	y	z	{		}	~	.	±	∅
130	△	Δ	θ							
140										
150										
160		ı	ϕ	£	¤	¥	¦	§	-	©
170	ª	«	¬	-	®	¯	°	±	²	³
180	´	µ	¶	·	¸	¹	º	»	¼	½
190	¾	¿	À	Á	Â	Ã	Ä	Å	Æ	Ç
200	È	É	Ê	Ë	Ì	Í	Î	Ï	Ð	Ñ
210	Ò	Ó	Ô	Õ	Ö	×	Ø	Ù	Ú	Û
220	Ü	Ý	Þ	ß	à	á	â	ã	ä	å
230	æ	ç	è	é	ê	ë	ì	í	î	ï
240	ð	ñ	ò	ó	ô	õ	ö	÷	ø	ù
250	ú	û	ü	ý	þ	ÿ				

Figure 5-14 Tmodel.fnt file

Extended ASCII Characters

Any ASCII character can be entered directly by entering the “\” character and the ASCII code. The letter A can be entered as \065. The primary use of the feature is to specify ASCII characters that are not easily entered directly from the keyboard such as the characters defined in the symbol font file.

In the P3server.ini file there is a setting to control the visibility of the “\” when editing ASCII characters over 127. When “Show8BitChars” is set to 1 editing text objects with AASCII characters over 127 will show the font character. When set to 0 the ASCII equivalent will be shown with the “\”. Ie. \176. This setting is established on installation and may only be changed by revising the p3server.ini file manually.

Working with Symbols

With **Terramodel** you have the ability to map symbols to point names as you label points so that all points having a common name (or common characters in their name) are labeled with the same symbol. You can define the symbol-to-point mapping in the ASCII text file SYMBOL.MAP, which can be modified by any text editor in the non-document mode.

Each line contains up to four fields, separated by commas:

```
field1,field2,field3[,field4]
```

where:

- field1 = Point name, up to eight characters long.
- field2 = Symbol number, corresponding to symbols contained in the SYMBOL.FNT file
- field3 = Symbol size, in sheet units defined by the **MeasUnit** command, drawn according to the current drawing scale.
- field4 = Optional. Where “A” is specified, the symbol size will be absolute (at the current default scale) rather than relative.

An example of a short and simple SYMBOL.MAP file is shown below:

Figure 5-15 Example Symbol Mapping File

```
BM, 46, .1
CB, 124, .1
CL, 38, .
COR, 50, .1
FH, 121, .1
IPF, 40, .1
IPS, 45, .1
MH, 48, 1.5, A
TP, 80, .1
WM, 56, .1
```



*All points with names not listed the SYMBOL.MAP file are labeled using the symbol number established in the **LabelPoints** command.*

You can find more information about symbol mapping in **DRAFT Menu**, “Label Points” in the On-Line Help system.

Working with Hatch Patterns

Hatching helps to clarify a drawing by making important project features stand out. You can hatch to highlight concrete surfaces, drainage basins, or grassed areas.

Once they are placed, hatch patterns become blocks in **Terramodel** that you can move, rotate, scale, and list just like any other **Terramodel** object. You can create hatching within the limits of selected closed boundaries (sets or polylines), and you can also select as boundaries any closed sets, polylines or text boxes that lie inside of the boundary to exclude them from hatching.¹ **Terramodel** places the hatch pattern on the current layer using the color assigned to the layer, and assigns it a name, *_hatch_nnn*, where *nnn* is a sequentially assigned number.

Where most CAD programs require a lot of memory for hatching, **Terramodel** stores the hatch pattern inside of each closed boundary in a compact format, conserving memory for other aspects of your project. If you want to modify a placed hatch block, you can “explode” it into its components and edit them, or re-create the hatch pattern definition and hatch the area again.²

Terramodel comes with several pre-defined hatch patterns. You can select one of these, create a pattern as you go, or design new patterns and store them in an external hatch pattern file. This file is accessible from any of your **Terramodel** projects.

Hatching with the Pattern Option

When you execute the **Hatch** command, **Terramodel** gives you a choice of hatching with a stored hatch pattern or creating a pattern as you go. The Pattern option opens the **HatchPat** command bar containing a hatch pattern control. You can use your up and down arrow keys on the keyboard to step through the available patterns.

¹ When you select multiple boundaries, **Terramodel** begins the hatch pattern when a hatch line first crosses a boundary line, and stops it when the hatch line crosses another boundary line. This is repeated each time a boundary line is crossed.

² Complex patterns can still consume a greater-than average amount of memory. If you explode a placed hatch block, then re-create it as a block, the block size (and memory required) will increase.

Hatch Pattern Dialog Box

Click the down arrow next to the hatch pattern control on the **HatchPat** command bar to open the *Hatch Pattern* dialog box. Click the name of the pattern you want to use, or click the Load button to load additional pattern definitions from a hatch pattern file.

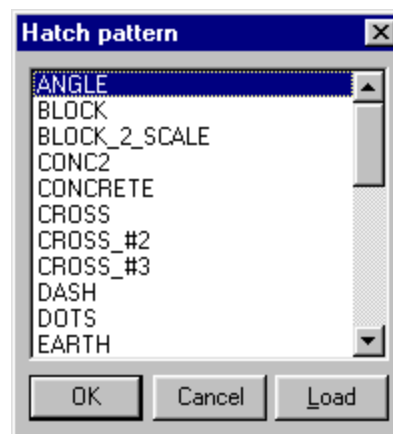


Figure 5-16 Hatch Pattern Dialog Box

The patterns that **Terramodel** provides are stored in TMODEL.PAT, which you can edit if you wish. You can also create your own hatch patterns and store them in a separate file. “Creating Custom Hatch Patterns”, on page 177, shows you how to store your own hatch patterns. **Terramodel** uses the same hatch pattern definition rules as AutoCAD. Additional hatch patterns in the public domain can be found on the internet.

Hatching with the User Option

The User option on the **Hatch** command bar (or the **HatchUser** command) creates an impromptu hatch pattern based on a given hatch angle, spacing, scale and boundaries. **Terramodel** uses the linetype of the current layer to create the hatching with this option, letting you vary your hatching effect.

Creating Custom Hatch Patterns

You can store your own hatch patterns in external ASCII files, and they will be accessible to any of your **Terramodel** project files. The format for hatch patterns is as follows:

```
*hatch name, comment
rotation angle,X-origin,Y-origin,delta X, delta Y, dash 1, dash 2,...
```

where,

- *hatch name* = the name of the hatch pattern. This name must always begin with an asterisk (*).
- comment* = a brief description of the hatch pattern.
- rotation angle* = a decimal polar angle where 00 is due East and 90 is due north.
- X-origin,Y-origin* = the X and Y origins of the first hatch line, measured with respect to the boundary line. Any value other than zero will cause the pattern to begin offset from the boundary line.
- delta X, delta Y* = the offset of each pattern line from the one before it. Offset distances are measured perpendicular to rotated lines.
- dash 1, dash 2,..* = dashed line pattern, where a negative value represents a pen-up movement, and a positive value indicates a pen-down movement. The pattern is repeated an infinite number of times using the rotation angle and origin until the boundary is intersected. If a dash pattern is not specified, a continuous line is drawn.

Your pattern definitions can also include comment lines, preceded by a semicolon (;). You can have an unlimited number of comments in your pattern definition file.

Each hatch pattern can have multiple line patterns. **Terramodel** repeats the patterns an infinite number of times until the boundary is filled.

Hatch Pattern Example

```
*BLOCK, Concrete block
00,0,0,0.0,0.2
90,0,0,0.2,0.2,0.2,-0.2
```

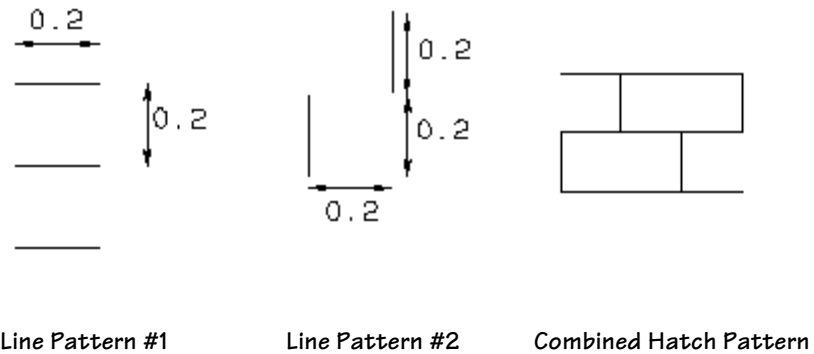


Figure 5-17 Block Hatch Pattern

Line pattern #1 creates continuous horizontal lines spaced .2 units apart. Line pattern #2 creates dashed, vertical lines that are offset 0.2 units in the x and y direction.

The effect is to create blocks 0.2 units tall and 0.4 units wide, which can be used for a typical 8 X 16 concrete block.

Working with Dynaviews

Dynaviews provide you with a way to display your project data in organized plot sheets. Each dynaview contains an image of original objects that will dynamically change as the objects change. Images in dynaviews can be rotated to any angle, while text inside them remains legible when you view it from the bottom or right-hand side of the sheet. You can show magnified details of particular project areas on the same sheet as an overall project plan, since each dynaview can maintain its own scale.

Dynaviews can be created in any view, as long as the view is of the plan, profile or sheet view *mode*.

To create a dynaview in the same view as the closed boundary:

- ▶ Use the **Pline**, **Box** or **PlotBox** commands to draw a closed boundary around the objects you want to display through the dynaview.
- ▶ Execute the **Dynaview** command. Enter a location in the view for placing the origin of the dynaview. A dynaview's origin is always the initial point on the closed boundary.
- ▶ Select the closed boundary, and enter the scale at which you want the dynaview shown. A scale of 1 displays the dynaview at the same scale as the original objects. A scale less than one displays a dynaview smaller than the original objects.
- ▶ Enter the angle at which you want the dynaview rotated. The rotation angle is measured clockwise with respect to the closed boundary.

To create a dynaview in a different view from the closed boundary:

- ▶ Use the **Pline**, **Box** or **PlotBox** commands to draw a closed boundary around the objects you want to display through the dynaview.
- ▶ Execute the **Dynaview** command. Open the view in which you want to create the dynaview.
- ▶ Enter a location in the view for placing the origin of the dynaview.
- ▶ Select the closed boundary, and enter the scale at which you want the dynaview shown.

When you create a dynaview in the plan and profile views, a scale of 1 displays the dynaview at the same scale as the original objects. If you are creating a dynaview in

the sheet view of objects in the plan or profile view, **Terramodel** will display the default scale as $1/(\text{current drawing scale of the original view})$. For instance, if the drawing scale of the plan view is $1"=50'$, the default scale for the dynaview will be 0.02. If you want to display the dynaview at a $1"=100'$ scale from the sheet view, you would enter 0.01 for the scale.

- Enter the angle at which you want the dynaview rotated.

To create plan/profile sheets using dynaviews:

- Use the **PlotBox** command to place a box around the objects in the plan view, and again to place a box around the profile view objects. Use the same width dimension for both boxes so that they will line up with each other when you place them together. Consider the total height of your sheet when entering the height dimensions for both boxes.
- In the sheet view, use the **Dynaview** command to place the dynaview of the plan. It is helpful, but not mandatory, to locate the origin of the dynaview by entering x- and y-positions.
- Use the **Dynaview** command again to place the dynaview of the profile.

When you enter a location, use the DxDy point snap option to place the origin at a specific x and y distance from the origin of the first dynaview. Make sure the Dx value is zero, so the two origins will line up horizontally, and the Dy value is the height of the profile box. The Dy value should be negative so that the profile dynaview will be placed below the plan dynaview. See chapter 6, **Using Terramodel Custom Controls**, “Object Snap Modes”, for more information on point snap options.

- Assuming the plan and profile views share the same horizontal scale, accept the default value for scale when you create both dynaviews. Do not rotate the dynaviews.

To add a named list of layers to a dynaview:

- Use the **LListSet** command to create a named list of layer to make visible through the dynaview.
- Use the **Edit** command to edit the dynaview, revise the layer list name as desired.

Additional add-ins

The default **Terramodel** installation will include many fonts, blocks and linetypes called additional add-ins or bonus files. These files include standard symbols to further automate the drafting process by placing north arrows, graphic scale bars, title blocks, plan and profile plates 1 and plate 2, and cross-section plate 3. Also included are dozens of linetypes. Descriptions and pictures of the blocks and linetypes, plus the instructions to operate them, are shown on the following pages.

Title Blocks

The default **Terramodel** installation will include additional title blocks. The title blocks included are listed below. These title blocks will construct a standard title sheet at various sizes. The title sheets are external blocks and should be placed in the sheet viewing mode. Use the **Block** command Insert option to place the block. The ATITLE through ETITLE blocks are set up at a set scale and should be placed at a scale of one.

The bonus files also include standard plate sheets for cross sections and plan and profiles. These sheets are set up for a 24"x36" sheet of paper. They should be placed with the scale set to one.

Block name	Sheet size
ATITLE	8 1/2"x11"
BTITLE	11"x17"
CTITLE	18"x24"
DTITLE	24"x36"
ETITLE	36"x48"
TITLE	Various sizes
PLATE1	24"x36" plan and profile
PLATE2	24"x36" plan and profile
PLATE3	24"x36" cross section

North Arrow Blocks

The default **Terramodel** installation will include additional North Arrow blocks. The north arrow blocks will construct arrows as shown below. The north arrows are external blocks and should be placed in the plan viewing mode and rotated as part of a dynaview in the sheet viewing mode. Use the **Block** command Insert option to place the north arrow of your choice. For the scale, input the scale of your drawing file. This will place a north arrow that is 3" high.

Before reading in the north arrows, you should use the **Layer** command, and set your current layer to the layer you wish to have the arrow placed on.

The north arrow blocks are shown below:

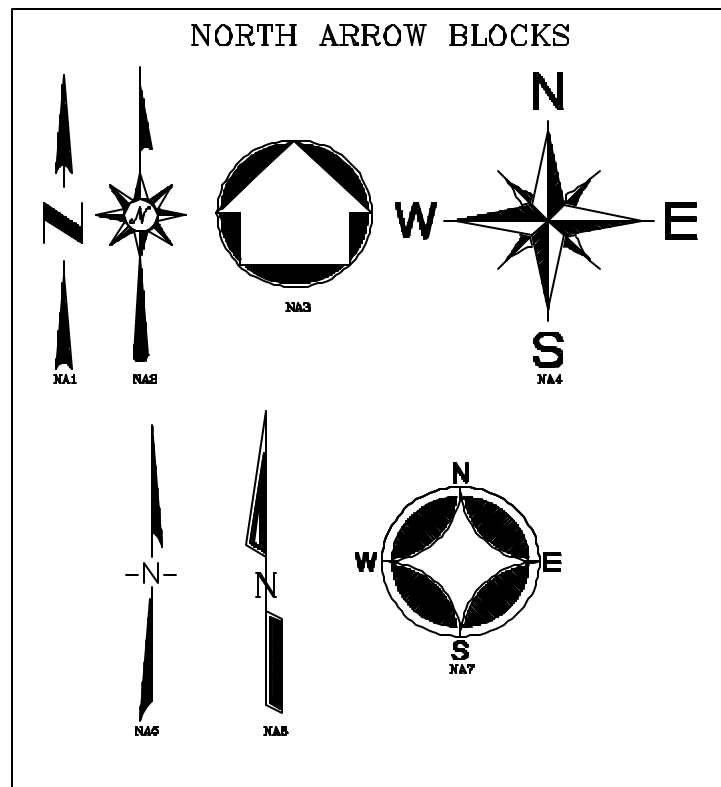


Figure 5-18 North Arrow Blocks

Graphic Scale Blocks

The default **Terramodel** installation will include additional graphic scale blocks. The graphic scale blocks will construct typical graphic scales. The scale blocks should be inserted in the sheet viewing mode. Use the **Block** command Insert option to place the graphic scale of your choice. When using in/feet based blocks for the scale enter one (1). This will place a scale that is 4" long and approximately 0.6" high. Metric blocks have been created based on sheet units of mm and should be inserted at a scale of 1 when using sheet units of mm.

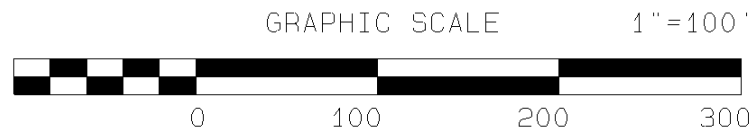


Figure 5-19 *Graphic Scale*

A complete list of the graphic scales included with the bonus files is shown below:

Block	Scale
5SCL	1"=5'
10SCL	1"=10'
20SCL	1"=20'
30SCL	1"=30'
40SCL	1"=40'
50SCL	1"=50'
100SCL	1"=100'
200SCL	1"=200'
300SCL	1"=300'
400SCL	1"=400'
500SCL	1"=500'
1000SCL	1"=1000'
M1SCL	1=1
M2_5SCL	1=2.5
M5SCL	1=5
M10SCL	1=10
M20SCL	1=20
M25SCL	1=25
M50SCL	1=50
M75SCL	1=75
M100SCL	1=100
M200SCL	1=200
M250SCL	1=250
M500SCL	1=500
M1000SCL	1=1000
M2500SCL	1=2500

Linetypes

The default **Terramodel** installation will include nine additional linetypes that can be used in **Terramodel** for special types of line patterns for your design work. You must load the linetypes from each file whenever you want to use a specific linetype or add them to your prototype project file. You could also load them all into your prototype project, but this would make over 200 linetypes to choose from. It is suggested that you created a prototype project that includes only the linetypes you might commonly use and then load additional linetypes only when required.



Please make a backup copy of TMODEL.LIN before you try adding any linetypes to it. Save the file and then load Terramodel .

To load the linetype file you have created (or one of the bonus files, if you choose to just load and use them as needed), use the **LineTypeSet** command.

The linetypes included with the bonus files are listed below. Pictures of these linetypes are included after the lists.

Linetype File Name: CURB.LIN

CURB.04R	Curb & Gutter	Gutter = 0.04" Offset Right; Dashes = 0.1"; Spacing = 0.1"
CURB.04L	Curb & Gutter	Gutter = 0.04" Offset Left; Dashes = 0.1"; Spacing = 0.1"
CURB.05R	Curb & Gutter	Gutter = 0.05" Offset Right; Dashes = 0.1"; Spacing = 0.1"
CURB.05L	Curb & Gutter	Gutter = 0.05" Offset Left; Dashes = 0.1"; Spacing = 0.1"
CURB.08R	Curb & Gutter	Gutter = 0.08" Offset Right; Dashes = 0.1"; Spacing = 0.1"
CURB.08L	Curb & Gutter	Gutter = 0.08" Offset Left; Dashes = 0.1"; Spacing = 0.1"
CURB.1R	Curb & Gutter	Gutter = 0.1" Offset Right; Dashes = 0.1"; Spacing = 0.1"
CURB.1L	Curb & Gutter	Gutter = 0.1" Offset Left; Dashes = 0.1"; Spacing = 0.1"
CURB.125R	Curb & Gutter	Gutter = 0.125" Offset Right; Dashes = 0.125"; Spacing = 0.125"
CURB.125L	Curb & Gutter	Gutter = 0.125" Offset Left; Dashes = 0.125"; Spacing = 0.125"
CURB.25R	Curb & Gutter	Gutter = 0.25" Offset Right; Dashes = 0.25"; Spacing = 0.25"
CURB.24L	Curb & Gutter	Gutter = 0.25" Offset Left; Dashes = 0.25"; Spacing = 0.25"

Linetype File Name: DISC.LIN

DISC.08	1.5" Centers and 0.08" High
DISC.1	1.5" Centers and 0.10" High
DISC.125	1.5" Centers and 0.125" High
DISC.25	1.5" Centers and 0.25" High

Linetype File Name: ELECTRIC.LIN

ELEC.08	E Symbol @ 1.5" Centers & 0.08" High
ELEC.1	E Symbol @ 1.5" Centers & 0.1" High
ELEC.125	E Symbol @ 1.5" Centers & 0.125" High
ELEC.25	E Symbol @ 3.0" Centers & 0.25" High
ELEC.08UE	UE Symbol @ 1.5" Centers & 0.08" High
ELEC.1UE	UE Symbol @ 1.5" Centers & 0.1" High
ELEC.125UE	UE Symbol @ 1.5" Centers & 0.125" High
ELEC.25UE	UE Symbol @ 3.0" Centers & 0.25" High
ELEC.08EP	Pole & EP Label @ 0.08" High
ELEC.1EP	Pole & EP Label @ 0.1" High
ELEC.125EP	Pole & EP Label @ 0.125" High
ELEC.25EP	Pole & EP Label @ 0.25" High
ELEC.08PP	Pole & PP Label @ 0.08" High
ELEC.1PP	Pole & PP Label @ 0.1" High
ELEC.125PP	Pole & PP Label @ 0.125" High
ELEC.25PP	Pole & PP Label @ 0.25" High
ELEC.08UCE	UEC Symbol @ 1.5" Centers & 0.08" High
ELEC.1UCE	UEC Symbol @ 1.5" Centers & 0.1" High
ELEC.125UCE	UEC Symbol @ 1.5" Centers & 0.125" High
ELEC.25UCE	UEC Symbol @ 3.0" Centers & 0.25" High
ELEC.08	Transmission Line @ 1.5" Centers and 0.08" High
ELEC.1	Transmission Line @ 1.5" Centers and .1" High
ELEC.125	Transmission Line @ 1.5" Centers and 1/8" High
ELEC.25	Transmission Line @ 3.0" Centers and 1/4" High

ELEC.08UG&P	UG Symbol & P Symbol @ 1.5" Centers and 0.08" High
ELEC.1UG&P	UG Symbol & P Symbol @ 1.5" Centers and 0.1" High
ELEC.125UG&P	UG Symbol & P Symbol @ 1.5" Centers and 0.125" High
ELEC.25UG&P	UG Symbol & P Symbol @ 3.0" Centers and 0.25" High
ELEC.08UG&P&T	UG symbol, P symbol, & T symbol @ 1.5" Centers and 0.08" High
ELEC.1UG&P&T	UG symbol, P symbol, & T symbol @ 1.5" Centers and 0.1" High
ELEC.125UG&P&T	UG symbol, P symbol, & T symbol @ 1.5" Centers and 0.125" High
ELEC.25UG&P&T	UG symbol, P symbol, & T symbol @ 3.0" Centers and 0.25" High

Linetype File Name: GROUND.LIN

GRND1.08L	Symbols on 0.54" Centers; Offset Left; 0.08" High
GRND1.08R	Symbols on 0.54" Centers; Offset Right; 0.08" High
GRND2.1L	Symbols on 0.675" Centers; Offset Left; 0.1" High
GRND2.1R	Symbols on 0.675" Centers; Offset Right; 0.1" High
GRND2.125L	Symbols on 0.8437" Centers; Offset Left; 0.125" High
GRND3.125R	Symbols on 0.8437" Centers; Offset Right; 0.125" High
GRND4.1L	Symbols on 1.6875" Centers; Offset Left; 0.25" High
GRND4.1R	Symbols on 1.6875" Centers; Offset Right; 0.25" High
GRND5.08L	Symbols on 0.15" Centers; Offset Left; 0.08" High
GRND5.08R	Symbols on 0.15" Centers; Offset Right; 0.08" High
GRND6.1L	Symbols on 0.2" Centers; Offset Left; 0.1" High
GRND6.1R	Symbols on 0.2" Centers; Offset Right; 0.1" High
GRND7.125L	Symbols on 0.25" Centers; Offset Left; 0.125" High
GRND7.125R	Symbols on 0.25" Centers; Offset Right; 0.125" High
GRND8.25L	Symbols on 0.375" Centers; Offset Left; 0.25" High
GRND8.25R	Symbols on 0.375" Centers; Offset Right; 0.25" High

Linetype File Name: PHONE.LIN

PHONE.08T	T Symbol @ 1.5" Centers & 0.08" High
PHONE.1T	T Symbol @ 1.5" Centers & 0.1" High
PHONE.125T	T Symbol @ 1.5" Centers & 0.125" High
PHONE.25T	T Symbol @ 3.0" Centers & 0.25" High
PHONE.08UT	UT Symbol @ 1.5" Centers & 0.08" High
PHONE.1UT	UT Symbol @ 1.5" Centers & 0.1" High
PHONE.125UT	UT Symbol @ 1.5" Centers & 0.125" High
PHONE.25UT	UT Symbol @ 3.0" Centers & 0.25" High
PHONE.08TP	Pole & TP Label @ 0.08" High
PHONE.1TP	Pole & TP Label @ 0.1" High
PHONE.125TP	Pole & TP Label @ 0.125" High
PHONE.25TP	Pole & TP Label @ 0.25" High
PHONE.08UC	UC Symbol @ 1.5" Centers & 0.08" High
PHONE.1UC	UC Symbol @ 1.5" Centers & 0.1" High
PHONE.125UC	UC Symbol @ 1.5" Centers & 0.125" High
PHONE.25UC	UC Symbol @ 3.0" Centers & 0.25" High

PHONE.08UTC	UTC Symbol @ 1.5" Centers & 0.08" High
PHONE.1UTC	UTC Symbol @ 1.5" Centers & 0.1" High
PHONE.125UTC	UTC Symbol @ 1.5" Centers & 0.125" High
PHONE.25UTC	UTC Symbol @ 3.0" Centers & 0.25" High
PHONE.08TVC	TVC Symbol @ 1.5" Centers & 0.08" High
PHONE.1TVC	TVC Symbol @ 1.5" Centers & 0.1" High
PHONE.125TVC	TVC Symbol @ 1.5" Centers & 0.125" High
PHONE.25TVC	TVC Symbol @ 3.0" Centers & 0.25" High
PHONE.08P&T	P Symbol & T Symbol @ 1.5" Centers & 0.08" High
PHONE.1P&T	P Symbol & T Symbol @ 1.5" Centers & 0.1" High
PHONE.125P&T	P Symbol & T Symbol @ 1.5" Centers & 0.125" High
PHONE.25P&T	P Symbol & T Symbol @ 3.0" Centers & 0.25" High
PHONE.08UG&T	UG Symbol & T Symbol @ 1.5" Centers and 0.08" High
PHONE.1UG&T	UG Symbol & T Symbol @ 1.5" Centers and 0.1" High
PHONE.125UG&T	UG Symbol & T Symbol @ 1.5" Centers and 0.125" High
PHONE.25UG&T	UG Symbol & T Symbol @ 3.0" Centers and 0.25" High

Linetype File Name: POLES.LIN

POLE.08OPEN	Poles As Circles @ 0.08" On The Line Points
POLE.1OPEN	Poles As Circles @ 0.10" On The Line Points
POLE.125OPEN	Poles As Circles @ 0.125" On The Line Points
POLE.25OPEN	Poles As Circles @ 0.25" On The Line Points
POLE.08SOLID	Poles As Solid-Fill Circles @ 0.08" On The Line Points
POLE.1SOLID	Poles As Solid-Fill Circles @ 0.10" On The Line Points
POLE.125SOLID	Poles As Solid-Fill Circles @ 0.125" On The Line Points
POLE.25SOLID	Poles As Solid-Fill Circles @ 0.25" On The Line Points
POLE.08P	Poles As Circles With A "P" Within @ 0.08" High On The Line Points
POLE.1P	Poles As Circles With A "P" Within @ 0.1" High On The Line Points
POLE.125P	Poles As Circles With A "P" Within @ 0.125" High On The Line Points
POLE.25P	Poles As Circles With A "P" Within @ 0.25" High On The Line Points
POLE.08U	Poles As Circles With A "U" Within @ 0.08" High On The Line Points

POLE.1U	Poles As Circles With A "U" Within @ 0.1" High On The Line Points
POLE.125U	Poles As Circles With A "U" Within @ 0.125" High On The Line Points
POLE.25U	Poles As Circles With A "U" Within @ 0.25" High On The Line Points
POLE.08T	Poles As Circles With A "T" Within @ 0.08" High On The Line Points
POLE.1T	Poles As Circles With A "T" Within @ 0.1" High On The Line Points
POLE.125T	Poles As Circles With A "T" Within @ 0.125" High On The Line Points
POLE.25T	Poles As Circles With A "T" Within @ 0.25" High On The Line Points
POLE.08E	Poles As Circles With An "E" Within @ 0.08" High On The Line Points
POLE.1E	Poles As Circles With An "E" Within @ 0.1" High On The Line Points
POLE.125E	Poles As Circles With An "E" Within @ 0.125" High On The Line Points
POLE.25E	Poles As Circles With An "E" Within @ 0.25" High On The Line Points

Linetype File Name: RR.LIN

RR.08DOUBLE	Double Railroad Line 0.08" Ties; 0.04" Between Tracks; 0.4" Tie-To-Tie
RR.1DOUBLE	Double Railroad Line 0.1" Ties; 0.05" Between Tracks; 0.5" Tie-To-Tie
RR.125DOUBLE	Double Railroad Line 0.125" Ties; 0.0625" Between Tracks; 0.625" Tie-To-Tie
RR.25DOUBLE	Double Railroad Line 0.25" Ties; 0.125" Between Tracks; 1.25" Tie-To-Tie
RR.08SINGLE	Single Railroad Line 0.08" Ties; 0.16" Tie-To-Tie
RR.1SINGLE	Single Railroad Line 0.1" Ties; 0.2" Tie-To-Tie
RR.125SINGLE	Single Railroad Line 0.125" Ties; 0.25" Tie-To-Tie
RR.25SINGLE	Single Railroad Line 0.25" Ties; 0.5" Tie-To-Tie

Linetype File Name: SLOPES.LIN

CUT.08	Cut Symbols 0.32" High	@ 0.5" Centers; Bases = 0.08" High
CUT.1	Cut Symbols 0.4" High	@ 0.625" Centers; Bases = 0.1" High
CUT.125	Cut Symbols 0.5" High	@ 0.78125" Centers; Bases = 0.125" High
CUT.25	Cut Symbols 1" High	@ 1.5625" Centers; Bases = 0.25" High
FILL.08	Fill Symbols 0.32" High	@ 0.5" Centers; Bases = 0.08" High
FILL.1	Fill Symbols 0.4" High	@ 0.625" Centers; Bases = 0.1" High
FILL.125	Fill Symbols 0.5" High	@ 0.78125" Centers; Bases = 0.125" High
FILL.25	Fill Symbols 1" High	@ 1.5625" Centers; Bases = 0.25" High
TB.08	TB Symbols 0.04" & 0.08" High	@ 0.2" Centers
TB.1	TB Symbols 0.05" & 0.1" High	@ 0.25" Centers
TB.125	TB Symbols 0.0625" & 0.125" High	@ 0.3125" Centers
TB.25	TB Symbols 0.125" & 0.25" High	@ 0.625" Centers

Linetype File Name: WATER.LIN

WATER.08W	"W" Symbols On 1.5" Centers; 0.08" High
WATER.1W	"W" Symbols On 1.5" Centers; 0.1" High
WATER.125W	"W" Symbols On 1.5" Centers; 0.125" High
WATER.25W	"W" Symbols On 3.0" Centers; 0.25" High
WATER.08W&1/2	"W" & 1/2" Symbols On 1.5" Centers; 0.08" High
WATER.1W&1/2	"W" & 1/2" Symbols On 1.5" Centers; 0.1" High
WATER.125W&1/2	"W" & 1/2" Symbols On 1.5" Centers; 0.125" High
WATER.25W&1/2	"W" & 1/2" Symbols On 3.0" Centers; 0.25" High
WATER.08W&3/4	"W" & 3/4" Symbols On 1.5" Centers; 0.08" High
WATER.1W&3/4	"W" & 3/4" Symbols On 1.5" Centers; 0.1" High
WATER.125W&3/4	"W" & 3/4" Symbols On 1.5" Centers; 0.125" High
WATER.25W&3/4	"W" & 3/4" Symbols On 3.0" Centers; 0.25" High
WATER.08W&1	"W" & 1" Symbols On 1.5" Centers; 0.08" High
WATER.1W&1	"W" & 1" Symbols On 1.5" Centers; 0.1" High
WATER.125W&1	"W" & 1" Symbols On 1.5" Centers; 0.125" High
WATER.25W&1	"W" & 1" Symbols On 3.0" Centers; 0.25" High

WATER.08W&2	"W" & 2" Symbols On 1.5" Centers; 0.08" High
WATER.1W&2	"W" & 2" Symbols On 1.5" Centers; 0.1" High
WATER.125W&2	"W" & 2" Symbols On 1.5" Centers; 0.125" High
WATER.25W&2	"W" & 2" Symbols On 3.0" Centers; 0.25" High
WATER.08W&4	"W" & 4" Symbols On 1.5" Centers; 0.08" High
WATER.1W&4	"W" & 4" Symbols On 1.5" Centers; 0.1" High
WATER.125W&4	"W" & 4" Symbols On 1.5" Centers; 0.125" High
WATER.25W&4	"W" & 4" Symbols On 3.0" Centers; 0.25" High
WATER.08W&6	"W" & 6" Symbols On 1.5" Centers; 0.08" High
WATER.1W&6	"W" & 6" Symbols On 1.5" Centers; 0.1" High
WATER.125W&6	"W" & 6" Symbols On 1.5" Centers; 0.125" High
WATER.25W&6	"W" & 6" Symbols On 3.0" Centers; 0.25" High
WATER.08W&8	"W" & 8" Symbols On 1.5" Centers; 0.08" High
WATER.1W&8	"W" & 8" Symbols On 1.5" Centers; 0.1" High
WATER.125W&8	"W" & 8" Symbols On 1.5" Centers; 0.125" High
WATER.25W&8	"W" & 8" Symbols On 3.0" Centers; 0.25" High
WATER.08W&10	"W" & 10" Symbols On 1.5" Centers; 0.08" High
WATER.1W&10	"W" & 10" Symbols On 1.5" Centers; 0.10" High
WATER.125W&10	"W" & 10" Symbols On 1.5" Centers; 0.125" High
WATER.25W&10	"W" & 10" Symbols On 3.0" Centers; 0.25" High
WATER.08W&12	"W" & 12" Symbols On 1.5" Centers; 0.08" High
WATER.1W&12	"W" & 12" Symbols On 1.5" Centers; 0.10" High
WATER.125W&12	"W" & 12" Symbols On 1.5" Centers; 0.125" High
WATER.25W&12	"W" & 12" Symbols On 3.0" Centers; 0.25" High
WATER.08W&14	"W" & 14" Symbols On 1.5" Centers; 0.08" High
WATER.1W&14	"W" & 14" Symbols On 1.5" Centers; 0.10" High
WATER.125W&14	"W" & 14" Symbols On 1.5" Centers; 0.125" High
WATER.25W&14	"W" & 14" Symbols On 3.0" Centers; 0.25" High
WATER.08W&16	"W" & 16" Symbols On 1.5" Centers; 0.08" High
WATER.1W&16	"W" & 16" Symbols On 1.5" Centers; 0.10" High
WATER.125W&16	"W" & 16" Symbols On 1.5" Centers; 0.125" High
WATER.25W&16	"W" & 16" Symbols On 3.0" Centers; 0.25" High

Examples of the linetypes are shown below.

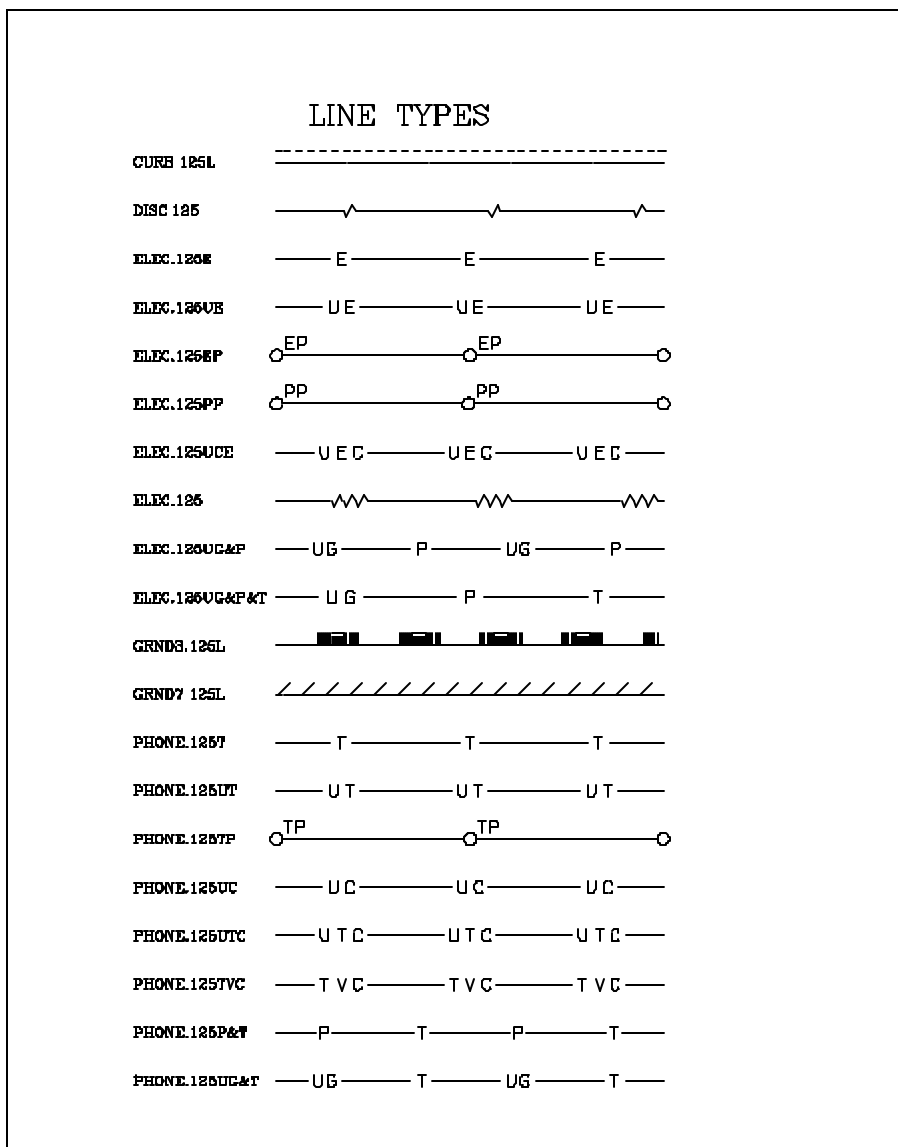


Figure 5-20 Example Line Types

Examples continued.

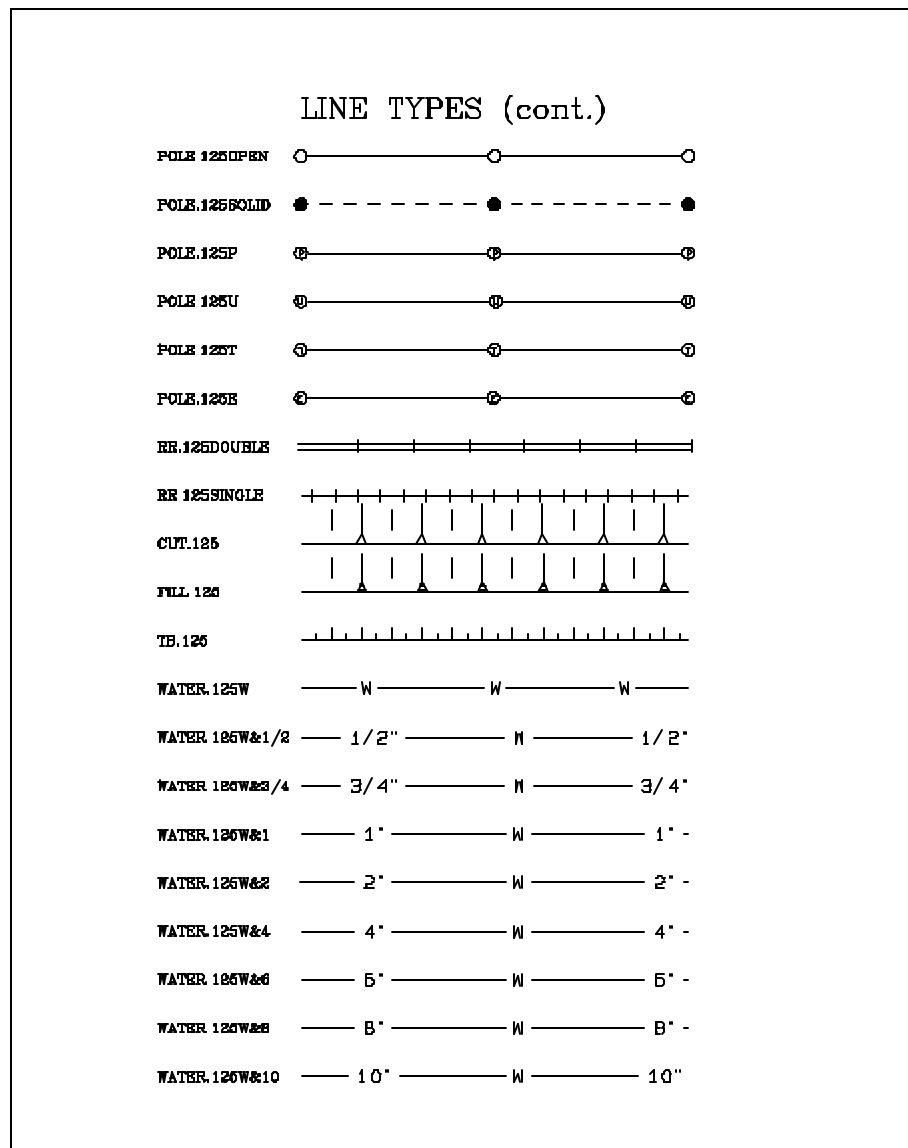


Figure 5-21 Example Line Types

Examples continued.

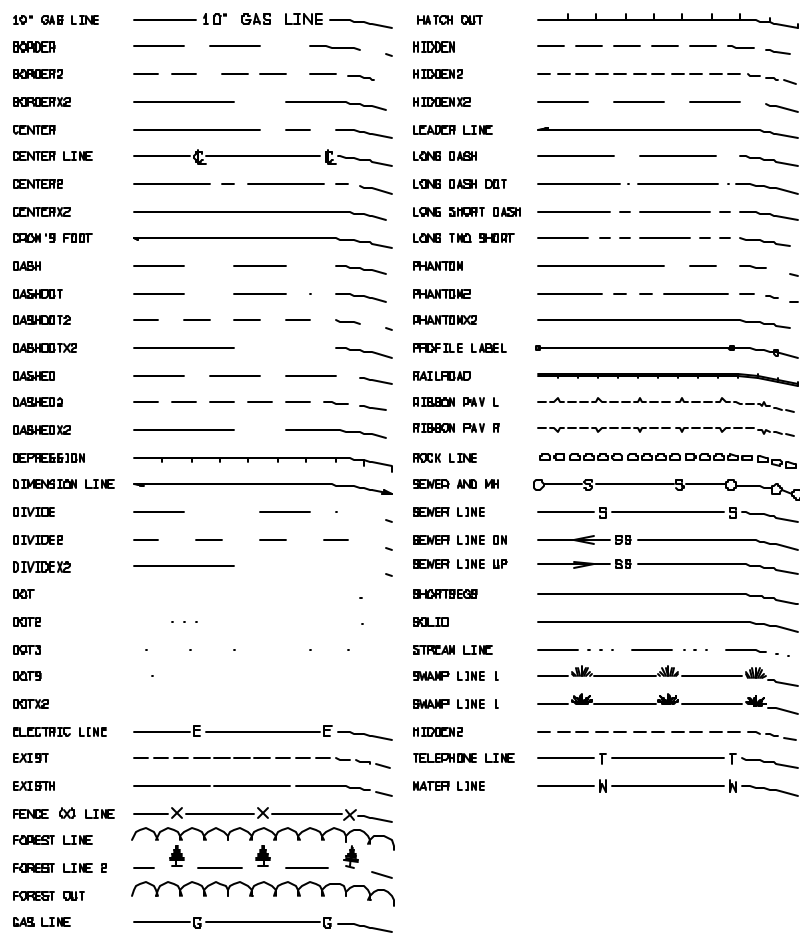


Figure 5-22 Tmodel.lin Line Types


Fonts

The default **Terramodel** installation will include several fonts in addition to basic TMODEL, TMODEL and the SYMBOL fonts. Some of these fonts are clones of the AutoCAD fonts and are licensed from Dave Edwards Consulting/TC Fonts (334.221.7919). TC Fonts has hundreds of fonts that can be purchased for use with Terramodel or AutoCAD. Visit www.tcfonds.com to view the complete selection of fonts. Be sure to tell Dave you are a Terramodel user.

The .shp format of these fonts are not automatically installed. These files may be copied from the shpfiles folder on the Terramodel CD but are not normally needed. The AutoCAD style format of these fonts are placed in the *installation folder\shared\fonts\acad_files* folder. These fonts should be moved to a location where they can be used by AutoCAD. The default.acf AutoCAD import/export configuration file has maps the native AutoCAD font names to the clone font names.

TC-COMP **Sphinx of Black Quartz, Judge My Vow!**
 TC-GDT SⓅⓂ≡∅∇ □// BⓁ∠□∩ Q—∠ⓄⓂⓂⓂ
 TC-GOTHE **Sphinx of Black Quartz, Judge My Vow!**
 TC-GOTHG **Sphinx of Black Quartz, Judge My Vow!**
 TC-GOTHI **Sphinx of Black Quartz, Judge My Vow!**
 TC-GREKC Σπηινξ οφ Βλαχκ Θυαρτζ, ουδγε Μψ Vow!
 TC-GREK5 Σπηινξ οφ Βλαχκ Θυαρτζ, ουδγε Μψ Vow!
 TC-ISQP1 Sphinx of Black Quartz, Judge My Vow!
 TC-ISQP2 Sphinx of Black Quartz, Judge My Vow!
 TC-ISQP3 Sphinx of Black Quartz, Judge My Vow!
 TC-ISQT1 Sphinx of Black Quartz, Judge My Vow!
 TC-ISQT2 Sphinx of Black Quartz, Judge My Vow!
 TC-ISQT3 Sphinx of Black Quartz, Judge My Vow!
 TC-ITAL *Sphinx of Black Quartz, Judge My Vow!*
 TC-ITALC *Sphinx of Black Quartz, Judge My Vow!*
 TC-ITALT *Sphinx of Black Quartz, Judge My Vow!*
 TC-MQND Sphinx of Black Quartz, Judge My Vow!
 TC-RDMNC **Sphinx of Black Quartz, Judge My Vow!**
 TC-RDMND **Sphinx of Black Quartz, Judge My Vow!**
 TC-RDMNS Sphinx of Black Quartz, Judge My Vow!
 TC-ROMANT **Sphinx of Black Quartz, Judge My Vow!**
 TC-SCRPC *Sphinx of Black Quartz, Judge My Vow!*
 TC-SCRPS *Sphinx of Black Quartz, Judge My Vow!*
 TC-SIMP Sphinx of Black Quartz, Judge My Vow!
 TC-TXT Sphinx of Black Quartz, Judge My Vow!

Figure 5-24 AutoCAD Clone Font Files

TMOBELF *Sphinx of Black Quartz, Judge My Vow!*
 TMOBELL *Sphinx of Black Quartz, Judge My Vow!*
 SYMBOL 

AMTYPENH **Sphinx of Black Quartz, Judge My Vow!**
 AMTYPENL **Sphinx of Black Quartz, Judge My Vow!**
 AMTYPENO **Sphinx of Black Quartz, Judge My Vow!**
 ARCHCHIS *Sphinx of Black Quartz, Judge My Vow!*
 ARCHPRES *Sphinx of Black Quartz, Judge My Vow!*
 ARCHQUIK *Sphinx of Black Quartz, Judge My Vow!*
 ARCHSING *Sphinx of Black Quartz, Judge My Vow!*
 AVONBRNH *Sphinx of Black Quartz, Judge My Vow!*
 AVONBRNL *Sphinx of Black Quartz, Judge My Vow!*
 AVONBRNO *Sphinx of Black Quartz, Judge My Vow!*
 BOLONYBH **Sphinx of Black Quartz, Judge My Vow!**
 BOLDNYBL **Sphinx of Black Quartz, Judge My Vow!**
 BOLDNYBO *Sphinx of Black Quartz, Judge My Vow!*
 BOLONYHH **Sphinx of Black Quartz, Judge My Vow!**
 BOLONYHL **Sphinx of Black Quartz, Judge My Vow!**
 BOLONYHO *Sphinx of Black Quartz, Judge My Vow!*
 MULTPLNH **Sphinx of Black Quartz, Judge My Vow!**
 MULTPLNL **Sphinx of Black Quartz, Judge My Vow!**
 MULTPLNO *Sphinx of Black Quartz, Judge My Vow!*
 PEANUTNH *Sphinx of Black Quartz, Judge My Vow!*
 PEANUTNL *Sphinx of Black Quartz, Judge My Vow!*
 PEANUTNO *Sphinx of Black Quartz, Judge My Vow!*
 TECHNQH *Sphinx of Black Quartz, Judge My Vow!*
 TECHNQL *Sphinx of Black Quartz, Judge My Vow!*
 TECHNQN *Sphinx of Black Quartz, Judge My Vow!*
 VELVETBH **Sphinx of Black Quartz, Judge My Vow!**
 VELVETBL **Sphinx of Black Quartz, Judge My Vow!**
 VELVETBO *Sphinx of Black Quartz, Judge My Vow!*
 VELVETHH **Sphinx of Black Quartz, Judge My Vow!**
 VELVETHL **Sphinx of Black Quartz, Judge My Vow!**
 VELVETHO *Sphinx of Black Quartz, Judge My Vow!*
 VELVETNH **Sphinx of Black Quartz, Judge My Vow!**
 VELVETNL **Sphinx of Black Quartz, Judge My Vow!**
 VELVETNO *Sphinx of Black Quartz, Judge My Vow!*

Figure 5-25 Terramodel Font Files

Sample Blocks

Several sample block are installed with Terramodel, here are a few to try.

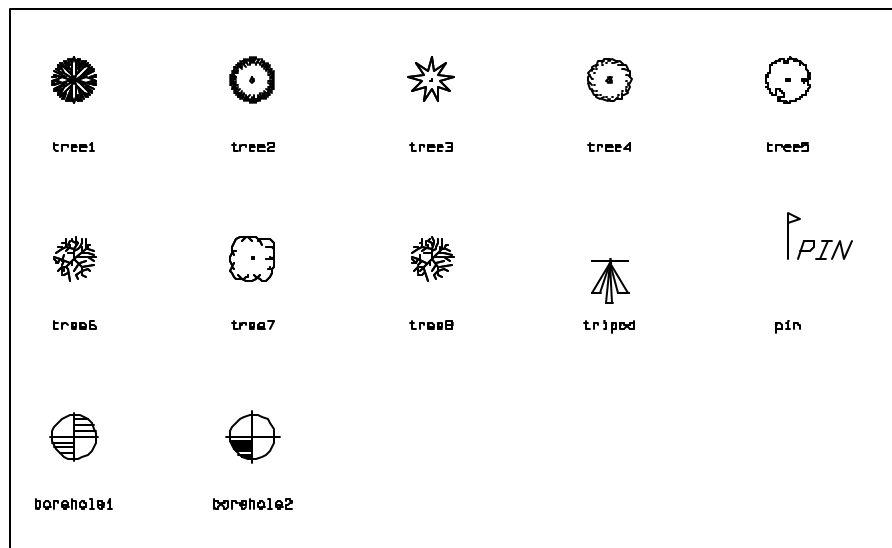


Figure 5-26 Terramodel Sample Blocks

Sign Blocks

The default **Terramodel** installation *does not* include the sign blocks. To use the sign blocks copy all of the files from the SIGNBLKS folder to your installation folder of **Terramodel**. The sign.pro project file shows all of the signs with the block names.






































GUIDE SIGNS						
 n1_1a	 n1_1b	 n1_2	 n1_3	 n1_4a	 n1_4b	 n1_6
 n1_5a	 n1_5b	 n1_7	 n2_1	 n3_1	 n3_2	 n3_3
 n3_4	 n4_1	 n4_1a	 n4_5	 n4_6	 n4_7	 n4_8
 n6_11	 n6_1r	 n6_2l	 n6_2r	 n6_3l	 n6_3r	 n6_23
 n6_2r	 n6_3	 n6_4	 n6_5l	 n6_5r	 n6_6l	 n6_7l
					 n6_6r	 n6_7r

Figure 5-27 Bonus Sign Blocks

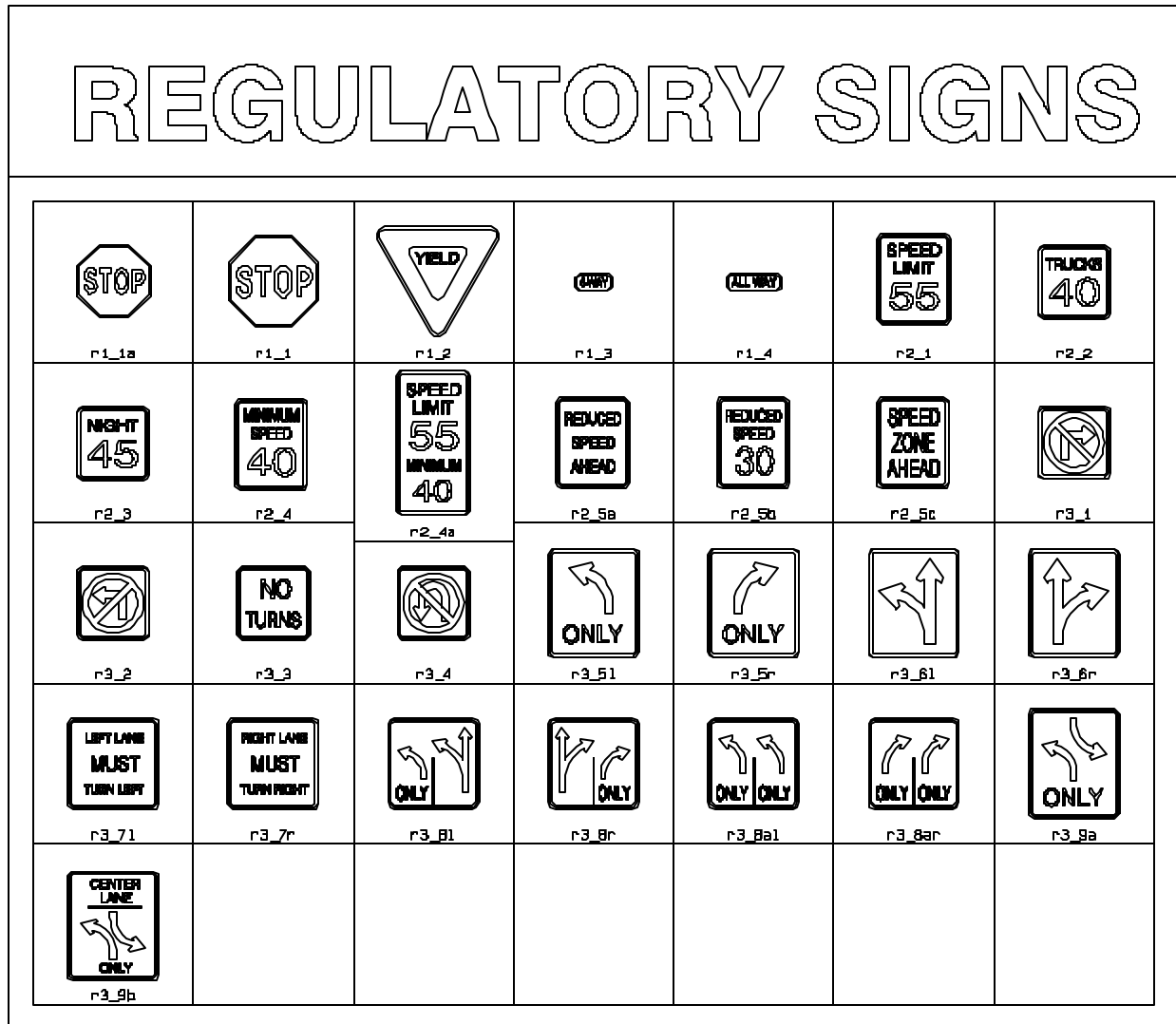


Figure 5-28 Bonus Sign Blocks













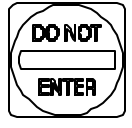















REGULATORY SIGNS						
 r4_1	 r4_2	 r4_3	 r4_5	 r4_5a	 r4_6	 r4_7
 r4_7b	 r4_7a	 r4_8	 r4_8a	 r4_8a	rx_x	rx_x
 r5_1	 r5_1a	 r5_2	 r5_3	 r5_4	 r5_5	r5_6
r5_7	r5_8	r5_9	 r5_10a	 r5_10b	 r5_10c	 r5_10d
 r6_11	 r6_1r	 r6_2l	 r6_2r	 r6_3	 r6_3a	rx_x

Figure 5-29 Bonus Sign Blocks

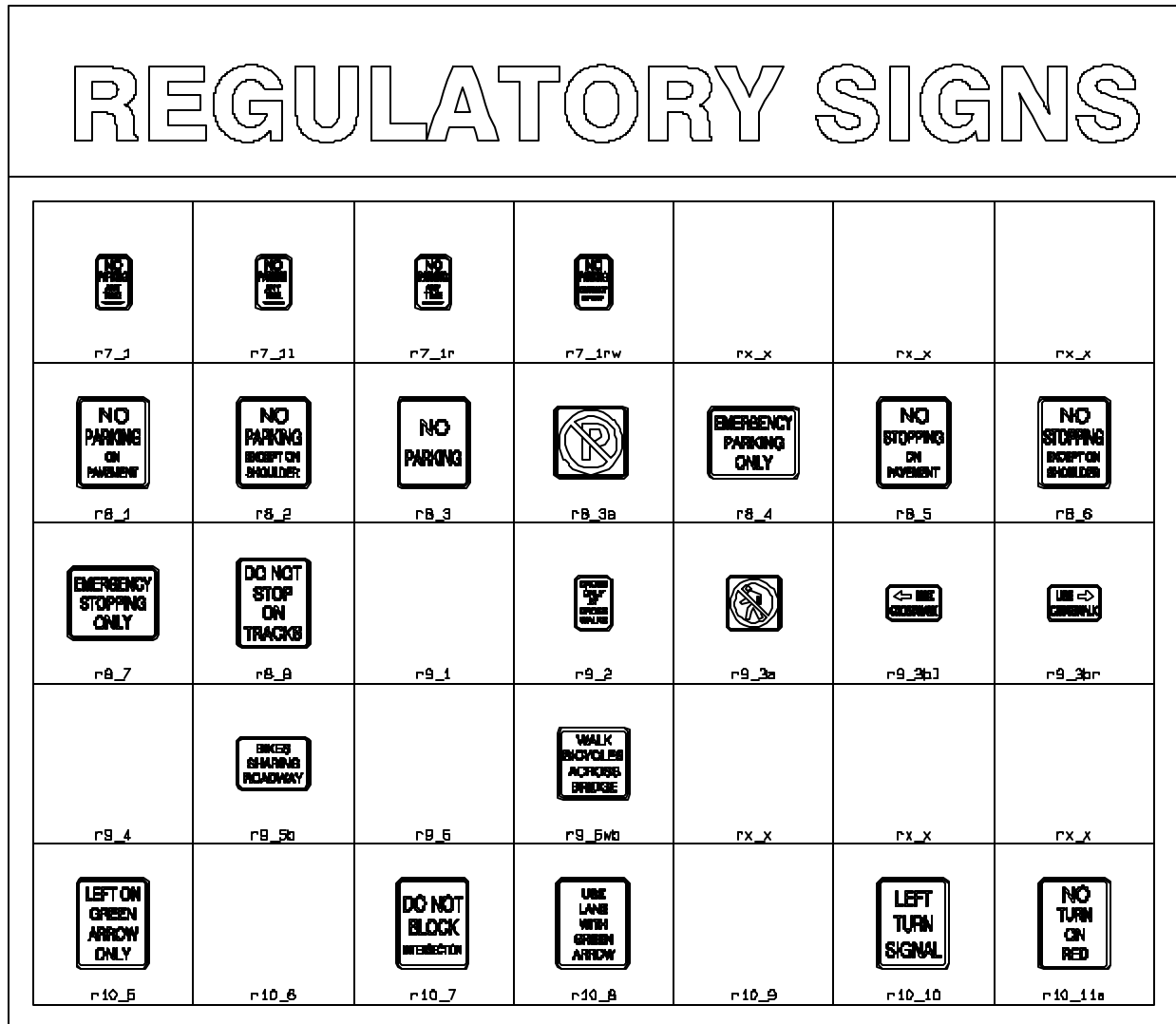


Figure 5-30 Bonus Sign Blocks



REGULATORY SIGNS						
						
r10_11b	r11_1	rx_x	rx_x	rx_x	rx_x	rx_x
rx_x	rx_x	rx_x	rx_x	rx_x	rx_x	rx_x
rx_x	rx_x	rx_x	rx_x	rx_x	rx_x	rx_x
rx_x	rx_x	rx_x	rx_x	rx_x	rx_x	rx_x
rx_x	rx_x	rx_x	rx_x	rx_x	rx_x	rx_x

Figure 5-31 Bonus Sign Blocks

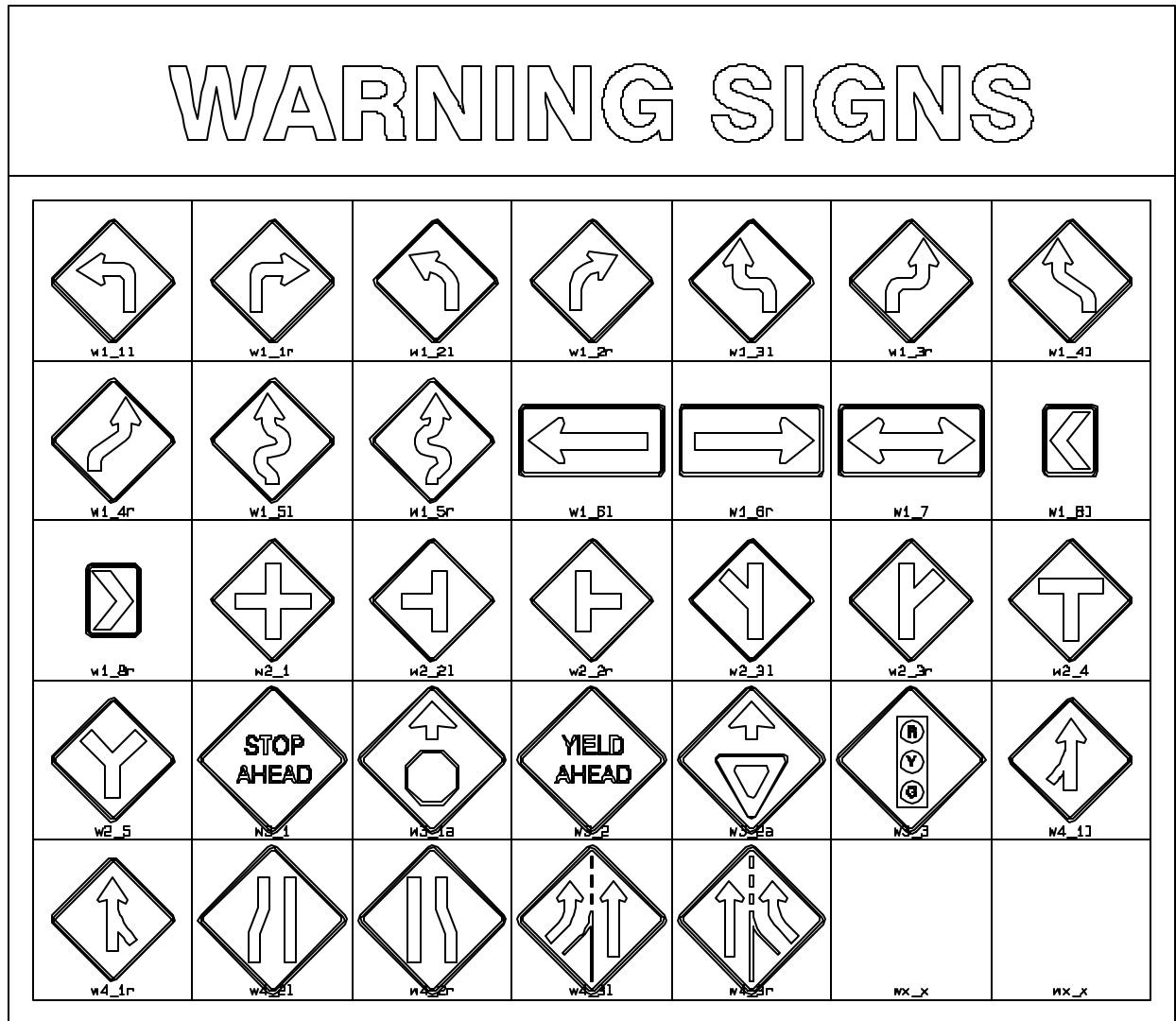


Figure 5-32 Bonus Sign Blocks

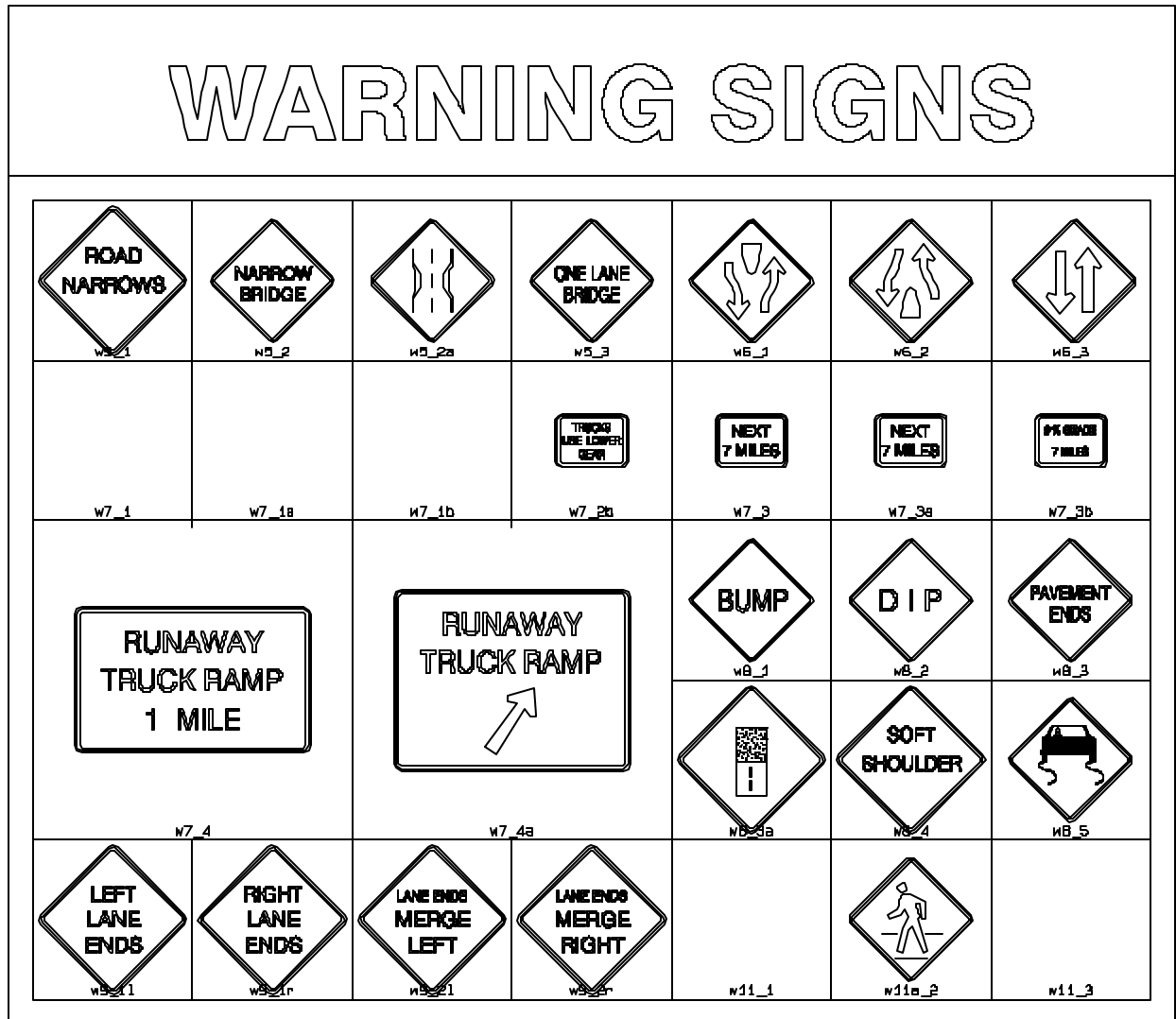


Figure 5-33 Bonus Sign Blocks

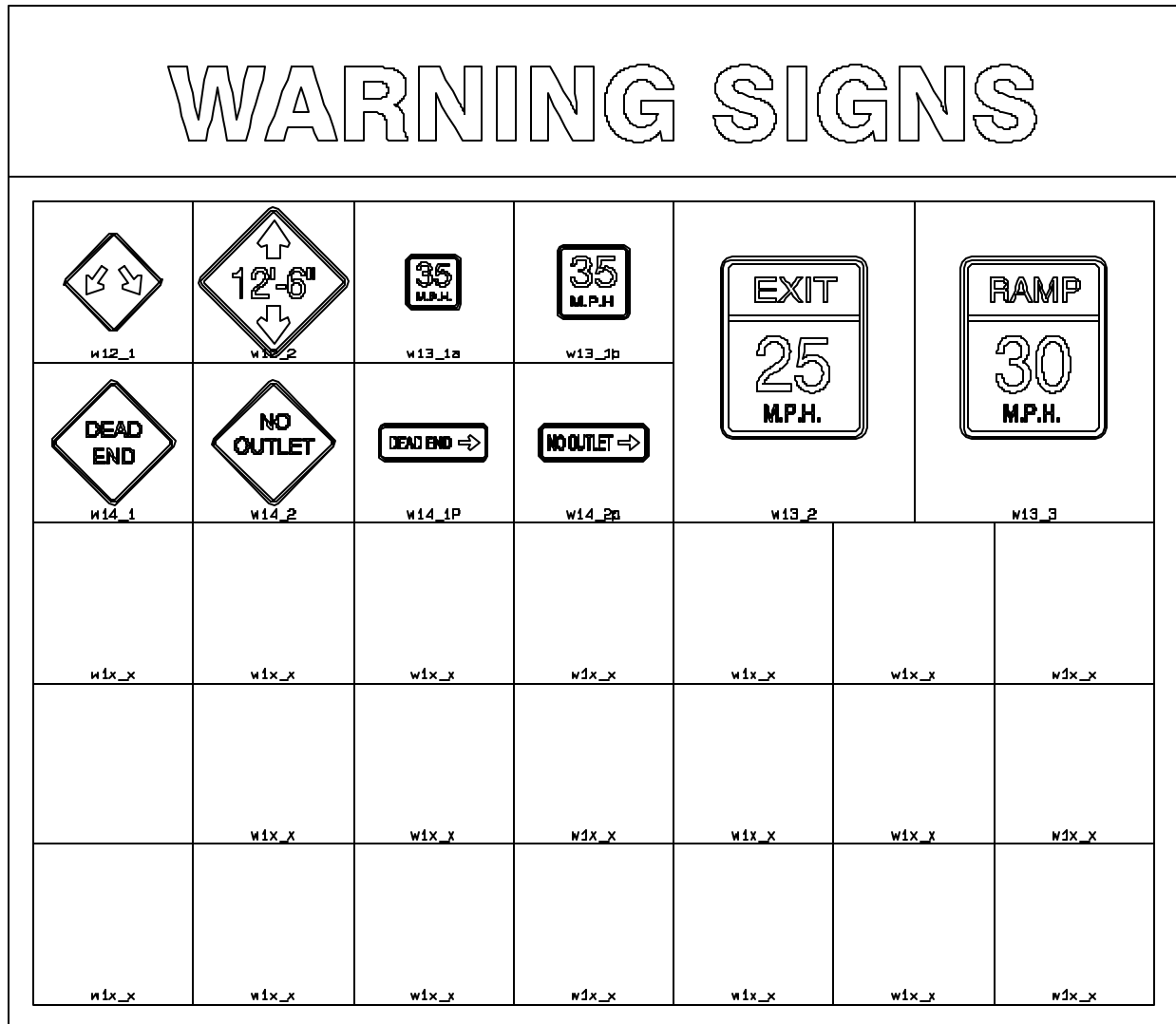


Figure 5-34 Bonus Sign Blocks

Chapter Six

Using Terramodel Custom Controls

Standard Windows control types provide a user interface for manipulating text and listed data that is consistent between applications. **Terramodel**, however, manipulates surveying, engineering and graphical data. Working efficiently with such data requires controls not found in most Windows applications. Fortunately, Windows provides a way for developers to add their own “custom” controls.

This chapter tells you how to use the custom controls we have provided to help you manipulate this complex data. Each control type was carefully designed to process a specific form of data, to be as intuitive as possible, and to be consistent as possible with standard controls and with other custom controls. Once you have mastered these controls, **Terramodel** will be as predictable and easy to understand as any other Windows application.

So that you will know how to respond to each prompt in a **Terramodel** command, the custom control type in use is always displayed in the right-hand field of the Status bar. As you move from one prompt (or input field) to another, watch the Status bar to see which control is active.

CAD Controls

Color Control



This control is used to select a color number. You will see the color control whenever you have to select a color. It appears in the **Color** command, in the Point and Object color controls on the toolbar, as well as in dialog boxes where you select a color for layers, text, etc. When this control is active, you will see *Color* in the lower right corner of the Status bar.

When the color control is in a dialog box, pressing the space bar, clicking the down arrow on the control, or double-clicking within the control opens the *Color Selection* dialog box. This dialog box displays a range of 64 color numbers at a time with corresponding colors. By selecting the range radio button you can view up to four ranges of color numbers (up to 255 color numbers). The cursor left and cursor right keys change the color number up and down by one. The cursor up and down keys increment the color number by eight. You may also type in the desired color number.

When the color control is displayed in the Toolbar or on the command bar (as opposed to in a dialog box), it has one additional capability. When the control is active, you can pick any object on the graphics screen and the color number in the control will change to the color of the object you selected. To remove the focus from the control while in the Toolbar, simply press Esc.

The actual color assigned to each color number is controlled by the **Palette** command. For more information about the color palette, see chapter 5, *Understanding Terramodel CAD*, “Working With Colors”.



When an object is assigned a color of zero, it assumes the color assigned to the layer on which it resides.

Edit Control

This control is used to specify a text string. You will see the edit control in commands where you have to enter text, such as naming objects, layers, or linetypes, or when you assign names to tables, blocks or hatch patterns. When this control is active, you will see *Edit* in the lower right corner of the Status bar.

You can enter any alphanumeric value within the maximum text length supported by the command you are using.

Layer Control



This control is used to select a layer or create a new layer. You will see the layer control whenever you have to select a layer. It appears in the **LSet** command, in the layer control on the Toolbar, as well as in dialog boxes where you select a layer for storing text, polylines, and other objects. When this control is active, you will see *Layer* in the lower right corner of the Status bar.

When the control is in a dialog box, pressing the space bar, clicking the down arrow on the control, or double-clicking within the control display opens the *Layer Selection* dialog box, with a full list of names of the project's layers. When the layer control is active you can use the up, down, left and right arrow keys on your keyboard to cycle through the names of the layers in your project file. You may also type a character and the list will move to the first layer name beginning with that character. Pressing the same character again will go to the next name in the list beginning with that character. The New command button of the *Layer Selection* dialog box lets you to create a new layer and define its colors, linetypes, and other properties.

When the layer control is used from the Toolbar or from the command bar during a command (as opposed to in a dialog box), the control has one additional capability. When it is active, you can pick any object on the graphics screen and the layer name in the control will change to the layer of the object you selected. To remove the focus from the control while in the toolbar, press Esc.

For more information about using layers, see chapter 5, *Understanding Terramodel CAD*, "Working With Layers".



Layer zero is reserved for special use by the program and shouldn't be used to store objects.

Linetype Control



This control is used to select a linetype or load a linetype file. You will see the linetype control whenever you have to select a linetype. It appears in the linetype control on the Toolbar, as well as in dialog boxes where you select a linetype for sets and polylines. When this control is active, you will see *Linetype* in the lower right corner of the Status bar.

When the control is in a dialog box, pressing the space bar, clicking the down arrow on the control, or double-clicking within the control display opens the *Linetype* dialog box, with a full list of names of the available linetypes for your project file. When the linetype control is active you can use the up, down, left and right arrow keys on your keyboard to cycle through the names of the linetypes in your project file. You may also type a character and the list will move to the first linetype name beginning with that character. Pressing the same character again will go to the next name in the list beginning with that character. The Load command button of the *Linetype* dialog box lets you load additional linetypes.

When the linetype control is used from the Toolbar or from the command bar during a command (as opposed to in a dialog box) this control has one additional capability. When this control is active, you can pick any object on the graphics screen and the linetype name in the control will change to the linetype of the object you selected. To remove the focus from the control while in the toolbar, press Esc.

For more information on linetypes, see chapter 5, *Understanding Terramodel CAD*, “Working With Linetypes”.

Hatch (Pattern) Control



This control is used to specify a hatch pattern or to load a hatch pattern definition file. When this control is active, you will see *Hatch* in the lower right corner of the Status bar.

When the control is in a dialog box, pressing the space bar, clicking the down arrow on the control or double-clicking within the control display opens the *Hatch pattern* dialog box, with a full list of names of the available hatch patterns for your project file. When the hatch control is active you can use the up, down, left and right arrow keys on your keyboard to cycle through the names of the patterns available. You may also type a character and the list will move to the first pattern name beginning with that character. Pressing the same character again will go to the next name in the list beginning with that character. The Load command button of the *Hatch pattern* dialog box lets you load additional patterns.

When the hatch control is used from the command bar during a command (as opposed to in a dialog box) this control has one additional capability. When this control is active, you can pick any hatch pattern (a block is object type) on the graphics screen and the pattern name in the control will change to the pattern of the block you selected.

For more information on hatch patterns, see chapter 5, *Understanding Terramodel CAD*, “Working With Hatch Patterns”.

Text Size Control

This control is used to specify text height. You will see the text size control in dialog boxes where you have to enter a text height, such as in the **LabelSets** command. When this control is active, you will see *TextSize* in the lower right corner of the Status bar.

This control operates similarly to the distance control except that you enter a text height value in the currently selected sheet units (inch, mm, cm, dm, m). Sheet units are selected from the **MeasUnit** command. When you define a height by picking a distance from the graphics screen, **Terramodel** translates the height to sheet units based on the current drawing scale of the current view.

Name Control

This control is used to specify the name of an object. Enter the name just as you would with an edit text control. When this control is active, you can pick any object on the graphics screen and the name in the control will change to the name of the object you selected.

Locate Controls

Locate (Object)

This control is used to locate a single object (**Terramodel** database record). You will see this control used to specify an object whenever you use **Edit Object**, and in most of the COGO commands where you are creating lines based on the location of other objects. It also appears in many of the Select and Osnap controls. When this control is active, you will see *Locate* in the lower right corner of the Status bar.

Depending on the types of objects acceptable to the command this control can be limited by the program to only search for the required types of objects. An example of this is when you are prompted to locate a point object. In this case you do not have to change the **search** settings. **Terramodel** automatically will only let you find a point object. Some commands use the locate control to enable you to locate a set or polyline. When either of these two object types can be used the **search** settings enable you to control which object type will be selected. In the event **search** is off for both object types **Terramodel** will search for both object types.

You can specify the object to be located by:

- Entering “r=” followed by the record number.
- Graphically selecting an object using any of the object snap modes.
- Entering a name. (This is case sensitive and searches in all view modes. In the event more than one object has the same name you will be warned “Multiple records found” and no object will be selected.)
- Press the right mouse button and use the *Browse* option described below.
- Typing a point id.
- Typing a point number.

Locate (Element)

If the command requires a set line or polyline segment or vertex, the record number and segment or vertex number will be displayed. When selecting an element you may enter the object's record number and segment or vertex number as follows:

$r = (\textit{the desired record number}); (\textit{the segment or vertex number}).$

Once you have selected a line object, you can use the up and down arrow keys on your keyboard to “walk” along the segments until you locate the segment you want to use.

Locate (Browse)

Frequently when selecting a particular object there will be more than one object on top of another. Whenever the locate control is used you may press the right mouse button and select the *Browse* option to select a specific object from a group. This option opens the *Browse* command bar which has an object locate control, next, previous, OK and cancel options. Place the cursor over the objects to be located and pick the objects. **Terramodel** will find all of the objects within the limits of the aperture box and highlight the first object found. The next and previous buttons can then be used to select any of the objects found. As each object is highlighted brief information about the object will be displayed in the status bar. Select the OK button to select the current object and return the original command. The **ID** command uses a locate control and can be used to try the browse functionality.

Left/Right Control

This control lets you indicate on which side of a line (Left/Right or Near/Far) a point should be created or a solution should be found. When this control is active, you will see *Left/Right* in the lower right corner of the Status bar. You can specify the direction by picking a location or the desired side of a line with your mouse, enter L or R, or toggle between the two choices using the up and down arrow keys.

This control appears in such commands as **BBear** or **BDist**.

Number Controls

Point Number Control

This control is used to specify a point number or point id. Commands that potentially create many points as a part of a design process do not allow you to directly assign a point number. These types of commands will always use the next available un-used point number beginning with the *Start point number* setting of the **PointSet** command.

This control is used by a few commands that create points but only when automatic point numbering is turned off in the **PointSet** command. When this control is active, you will see *PtNum* in the lower right corner of the Status bar.

You must enter a valid point number (a positive integer) or point id (unique alphanumeric id 16 characters or less) to be used. When this control is on the command bar you may also specify a point number or id by picking any point from the display.

Alphanumeric point ids may only be used if they have been enabled by specifying a maximum number of alphapoints in the **System** command.

Integer Number Control

This control is used to specify an integer value (a positive or negative whole number), including zero. You will use this control with commands like **LabelPoint** and **LabelSet** where you specify a symbol number, or other commands where only a whole number is acceptable input. When this control is active, you will see *Long* in the lower right corner of the Status bar. You must type the value—screen picking is not available.

Real Number Control

This control is used to specify a real number value (integers and decimals). This control appears in most dialog boxes and command bars where a decimal entry value is acceptable. When this control is active, you will see *Real* in the lower right corner of the Status bar. You must type the value (screen picking is not available).

Coordinate Geometry Controls

Angle Control

This control is used to specify an angle. You will see this control in such commands as **Rotate** and **TextRotate**. When this control is active, you will see *Angle* in the lower right corner of the Status bar. A positive angle is clockwise.

To specify an angle:

Either:

- ▶ With the cursor “rubber banding”, pick a point to specify the angle graphically.

Or:

- ▶ Press the right mouse button to use any of the angle snap modes. Angle snap modes are described in the *Snap Modes* section later in this chapter.

Or:

- ▶ Enter the angle using the angle entry formats discussed in **SETTINGS Menu**, “Units Settings”, the **UnitsSet** command, in the On-line Help system. A positive angle is clockwise.

Bearing Control

This control is used to specify a bearing or azimuth. It appears in both command bars and dialog boxes like **Traverse** and **DtmGrid** Settings. When this control is active, you will see *Bearing* in the lower right corner of the Status bar.

To specify a bearing:

Either:

- ▶ You can specify a bearing graphically, using the default “Of line” bearing snap mode, by simply clicking on an object as follows. If a straight line segment is found within the pick box when you click, then the bearing of that line segment will be used. The direction of the bearing is specified by picking a point on the line segment closer to the end to which you want to the bearing to travel. If a curved line segment is found within the pick box, the bearing of the tangent to that curve at that point will be computed. If instead a text object is found within the pick box, the bearing at which the text object is located will be found. This allows you to use the bearing control to alignment text objects with other text objects. If you do not pick a line segment or a text object, the “Pt-Pt” bearing snap mode will be used instead and the cursor will “rubber band” from the selected point allowing you to select a second point with which to define the bearing.

Or:

- ▶ You can press the right mouse button to explicitly select from any of the supported bearing snap modes. Bearing snap modes are described in the *Snap Modes* section later in this chapter.

Or:

- ▶ You can enter a point number to point number notation (e.g. 1>2) to recall the bearing between two point numbers.

Or:

- ▶ You can enter the an azimuth angle, or a bearing using the bearing angle entry described below.

When this control is active you can use the cursor up and cursor down keys to add or subtract 90 degrees to the currently displayed bearing. This allows you to easily specify a perpendicular to a selected line segment bearing.

The **UnitsSet** command also sets the default mode used when no code is specified. Typical angle modes are show below.

Table 6–1 Horizontal Angle Modes

Code	Description	Examples
NA or 9/	North Azimuth	NA692732 or 692732NA or 9/692732
SA or 0/	South Azimuth	SA1591513 or 1591513SA or 0/1591513
AR or 5/	Angle Right	AR1591513 or 1591513AR or 5/1591513
AL or 6/	Angle Left	AL1591513 or 1591513AL or 6/1591513
DR or 7/	Deflection Right	DR1591513 or 1591513DR or 7/1591513
DL or 8/	Deflection Left	DL1591513 or 1591513DL or 8/1591513
NE or 1/	Northeast	N692732E or 692732NE or 1/692732
SE or 2/	Southeast	S692732E or 692732SE or 2/692732
SW or 3/	Southwest	S692732W or 692732SW or 3/692732
NW or 4/	Northwest	N692732W or 692732NW or 4/692732

Point (Coordinate) Control

This control is used to specify an X, Y, and optional Z coordinates of a location. When this control is active, you will see *Point* in the lower right corner of the Status bar. Typically the option will be *Loc.*:

To specify a point(coordinate location):

Either:

- ▶ Pick a position in the display to graphically specify a location. Assuming there are no running snap modes active, the default operation does not use Z.

Or:

- ▶ Enter a single number to specify the coordinates of a point object.

Or:

- ▶ Enter two or three numbers separated by a space or comma to specify a coordinate pair or triplet. Use this format:

N,E(,Z) or
N E (Z)

Examples:

27,42(,1) or
27 42 1 or
27 42 or
27,42

In the plan view these would normally represent N, E(,Z). In the profile view these would normally represent Station, and Elevation. This control honors the ordering of the X, and Y values as specified for the current view (see the **ViewSet** command). If Z is entered in the point control and there is also an elevation control for the same point the Z value will automatically be placed in the elevation control.

Or:

- ▶ Specify a relative distance from a previous point. Use this format:

@N,@E,@Z) or
@N @E @Z)

Examples:

@27,@42 or
@27 @42 or

The @ symbol is used to specify a relative distance and must be used before each coordinate axis entry. You can enter a relative distance for one axis and an absolute value for the other axes. If Z is entered in the point control and there is also an elevation control for the same point, the Z value will automatically be placed in the elevation control.

In any view with a vertical exaggeration you can substitute a grade (G) for the elevation when setting a point relative to a previous point. This can be used in conjunction with a relative (@S) station value. Valid entries for grade are: (% or P) for per cent of grade and (; or R) for ratio grade, e.g. 2:1.

Examples:

@500,2% or
@500 2P

In any view with a vertical exaggeration you can substitute a Maximum (M) depth for the station and grade (G) for the elevation when setting a point relative to a previous point. The maximum depth will be the change in depth (elevation) from the previous point. Valid entries for grade are: (% or P) for per cent of grade and (; or R) for ratio grade, e.g. 2:1. This type of entry is typically used in defining templates used with roads. Example:

5,2% or
5 2%

Or:

- Press the right mouse button to use any of the point snap modes. Point snap modes are described in the *Snap Modes* section later in this chapter.

Distance Control

This control is used to specify a distance. Typically this is the horizontal distance but when calculating 3D points this can also be a slope distance (SD). You will find this control in such commands as **DDist** and **BDist**. When this control is active, you will see *Distance* in the lower right corner of the Status bar.

To specify a distance:

Either:

- ▶ With the cursor “rubber banding”, pick a point to specify the distance graphically. The default mode also will pick the distance “of line”, i.e., if a line segment is found the length of the segment will be used as the distance.

Or:

- ▶ Enter point number to point number (e.g. 1>2) to recall the distance between two point numbers.

Or:

- ▶ Enter the distance directly or as a slope distance. Example:

100 *or*
100SD

Or:

- ▶ Press the right mouse button to use any of the distance snap modes. Distance snap modes are described in the *Snap Modes* section later in this chapter.

Whenever this control is active you may use an algebraic expression to calculate the distance. To preview the calculated value, use the Tab key to leave the control. The expression will be replaced with the calculated value. The following are valid operators:

- Subtraction
- + Addition
- / Division
- x Multiplication
- () Parenthesis can be used as required

Elevation Control

This control is used to specify an elevation. It appears in such commands as **Elevation**, and the *Edit Point* dialog box. When this control is active, you will see *Elevation* in the lower right corner of the Status bar.

To specify an elevation:

Either:

- ▶ With the cursor, pick an object to specify the elevation of the object. If no object is found *(2D) will be used. If the object is a set the elevation will be interpolated along the slope of the line based on the selected position along the line.

Or:

- ▶ Enter the elevation directly (e.g. 1000).

Or:

- ▶ Press the right mouse button to use any of the elevation snap modes. The elevation snap modes are described in the *Snap Modes* section later in this chapter.

Hide and Pick Control



This control appears inside dialog boxes next to controls where you may want to pick from the display. By clicking this control, you can “hide” the dialog box and pick or select from the display. Once you have completed picking, the dialog box reappears.

When this control is selected, you will see only the name of the dialog box (which is now hidden) in the lower right corner of the Status bar.

Offset Control

This control is used to specify an offset from a given alignment. When this control is active, you will see *Offset* in the lower right corner of the Status bar.

You can type an offset distance or specify a position on the graphics screen to place an offset distance in this control. **Terramodel** uses the perpendicular distance from the alignment to the specified position to calculate the offset distance.

You may also press the right mouse button to use the *at point* (point) snap mode or any of the distance snap modes. The *at point* option gives you the ability to use any of the point snap modes to specify the offset distance from a specific point to the alignment.

Station Control

This control is used to specify a station with respect to a selected (or active) alignment. When this control is active, you will see *Station* in the lower right corner of the Status bar.

You can type a station value or specify a position on the graphics screen. **Terramodel** calculates the station from the specified position to a perpendicular point along the alignment.

You can also press the right mouse button to use the *at point* (point) snap mode. The *at point* option gives you the ability to use any of the point snap modes to specify the station along the alignment at a specific point.

If you have a roadway defined and are using station equations, you may enter the station and the segment to specify the station. In order to use this option you must have defined a road job and have defined the station equations. If there is more than one road job that uses the alignment, only the first one found can be used. An alignment with two station equations has three segments, one before the first equation, one between the two equations and one after the second equation. To define the station and segment enter "(station):(segment)" e.g., 14000:3. If no segment is specified the first segment containing the specified station will be used.

You may also specify an absolute distance along the alignment regardless of station equations or beginning station by using "0" (zero) for the segment number, e.g., 14000:0.

Vertical (Angle) Control

This control is used to specify a vertical angle. When this control is active, you will see *Vertical* in the lower right corner of the Status bar.

To specify a vertical angle:

Either:

- ▶ Enter point number to point number (e.g. 1>2) to recall the vertical between two point numbers. The points must both have an elevation for this to determine a valid angle.

Or:

- ▶ Press the right mouse button to use any of the vertical point to point snap modes.

Or:

- ▶ Pick a line or two points in the profile view to specify the angle.

Or:

- ▶ Enter the vertical angle using the vertical angle entry formats discussed in **SETTINGS Menu**, “Units Settings”, **UnitsSet** command, in the On-line Help system. Example:

```
85 30 30Z
```

Remember that this command sets the default mode used when no code is specified. Typical angle modes are show below.

Table 6-2 Vertical Angle Modes

Code	Description	Examples
Z	Zenith 00 = straight up.	Z885413 or 885413Z
*	2D No vertical angle, a 2d point will be set.	
H	Horizontal 00 = level, horizontal.	H010547 or 010547H
N	Nadir 00 = straight down.	N910547 or 910547N
P or %	Percent slope % measured from horizontal.	P1.91, 1.91P, %1.91 or 1.91%
R	Ratio slope Horizontal run to vertical rise or vertical to horizontal based on the ConvSet command setting.	R52:1 or 52:1R
D	Distance Vertical distance or elevation difference between 2 points.	D3.67 or 3.67D

Select Controls

Select controls are controls that you use to select an object or objects to receive the effect of a command.

It is important to realize that a select control is always labeled according to the type of object(s) that you are allowed to select. For instance, it is labeled “Text” when you are allowed to select only text objects. In all cases, however, the lower right corner of the Status bar will have the word “Select”.

Building Selection Sets

The selection of objects is based on a group of criteria collectively known as a *selection expression*. You can create complex selection expressions by using select options to build a selection set.

Specify the select mode by using the cursor up/down keys or entering the first letter of the desired mode until the mode is shown in the select control. You may also press the right mouse button to display a menu to select the mode.

For most select modes you can immediately use the cursor to add to the selection set. Options that request a CAD property can be used by simply picking an object that has the desired CAD property. Options such as Color, Linetype, Elevation, Window remain the current mode and let you continue to add objects to the selection set. You may also click the button to the right of the control, or press the space bar, to open the corresponding dialog box or command bar for that option.

By combining the select options with the mathematical and logic expressions Equals, Or, And Not, Xor, And and Not, virtually any combination can be used to select objects.

Example:

```
Layer(points) and not Name(tree)
```

```
Color(12) or Color(13) and not Linetype(solid)
```

In addition to the select options that let you select using the cursor to select objects, and options that use dialog boxes, you can also use the Expression select option to type in an expression directly. While you build a selection set, you can use the Expression option at any time to view and edit your current expression.

Tips for Select options

The Selection Set options follow guidelines that, once you learn them, will help maximize your efficiency in using them.

- On and Off are used to toggle the visibility of objects. When objects are toggled off, they cannot be selected graphically, nor by the Window option, but all other select options will work.
- Unless otherwise stated, for most select options you can choose only objects in the current view or the view in which they were selected, unless you modify the expression directly.
- By naming your points and layers carefully, you can make selecting groups of objects by name much easier.



Objects are not selected until after you have finished building your selection set. Because you can access commands from within other commands, it is possible to create or delete objects inadvertently while building a selection set from within another command. This never causes Terramodel a problem but does require that the select expression be evaluated only when the command is completing its action.



Unless otherwise specified, selection is not restricted to the objects that are visible. Selecting objects is limited to the current view mode, view mode specified or view mode selected within, unless the expression is modified directly by the user. Objects that reside on layers that have their snap status off can not be graphically selected (pick, Window or Crossing options).

Choosing a Select Option

The select control is used whenever you want to select multiple objects. It lets you select one or more object types using many options. When the focus is on the select control, you will see the word *Select* in the right corner of the Status bar. (As shown in the example below, the select control is labeled “Objs” because almost any type of object can be selected.)

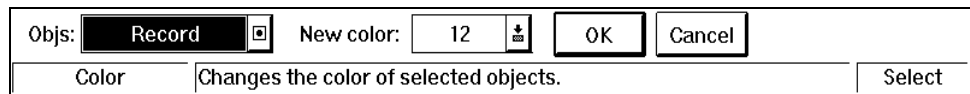


Figure 6-1 Sample Select Control
Command Bar

The initial select option that is shown is Rec (database record). As long as the select control is active, you can use the up and down arrow keys on your keyboard to change the select option. You can also click the right mouse button anywhere within the view window to display the select options, which are shown in the example below.

Color	Reference
Crossing	Type
Elev	View
Group	Window
Inside	Expression
Layer	Previous
Linetype	
Name	EQUALS =
NearLine	OR +
OfLine	AND NOT -
Points	XOR ^
Record	AND *
	NOT !

Figure 6-2 Select Options

Select Options

Color	Adds to the selection set objects that have a specified color. Pick an object to select by that object's color.
Crossing	Adds objects that are within or cross within the limits of a window defined by two opposite corners. Only visible objects are selected.
Elev	<p>Adds to the selection set objects that have a specified elevation or are within a specified elevation range.</p> <p>Click the button next to the select control to open the Elevation command bar. Enter the lower and upper limits of an elevation range. When the focus is on the control, you can also select an elevation by clicking an object that has the elevation you want to use.</p>
Group	<p>Adds to the selection set all objects having the same group code number.</p> <p>Enter the group code number of an object by picking the object on the graphics screen. Or, click the select control to open the Group command bar, where you enter an object's group code number. To assign a group code number to an object, use the Edit command and type in a group code (see EDIT Menu, "Edit Object" in the On-line Help system).</p>
Inside	Selects objects that fall entirely inside of a specified "closed" line. Terramodel adds all objects that lie within a selected line to the selection set. If you select a line that is not closed, Terramodel will assume the beginning and ending points are connected and select objects that lie within the line. Objects that extend outside of the line will not be selected. Only visible objects are selected.

Layer	<p>Adds to the selection set objects that are on a specified layer.</p> <p>When the focus is on the select control, you can select an object on the screen and Terramodel will add to the selection set all objects that reside on the layer of the object you selected. Or, click the button next to the control to open the <i>Select by Layer(s)</i> dialog box.</p>
Linetype	<p>Adds to the selection set objects that have a specified linetype.</p> <p>When the focus is on the select control, you can select an object on the screen and Terramodel will add to the selection set all objects that have the linetype of the object you selected. Or, click the button next to the control to open the <i>Select by Linetype</i> dialog box.</p>
Name	<p>Adds to the selection set objects that have a specified name. Note that name selection is case sensitive.</p>
NearLine	<p>Adds to the selection set objects that lie within a specified distance of a selected line.</p>
OfLine	<p>Adds points connected by a specified set line to the selection set.</p> <p>When the focus is on the select control you can pick a set line, or click the button next to the control and enter a set in the Offline command bar.</p>
Points	<p>Adds to the selection set a point having a specified number, or points within a point number range, using the <i>Select by Point Number</i> dialog box. This option does not support point ids (alphanumeric ids).</p>
Record	<p>Adds objects to the selection set by picking them one at a time or use the record number option.</p> <p>To use the record number option, press the space bar or click the button next to the select control to use the locate control. This lets you select by record number, point number or name. Only visible objects can be selected by picking, but any object can be selected by using the record number option.</p>

Reference	<p>Adds to the selection set objects that share an object reference number.</p> <p>Clicking the button next to the select control opens the Reference command bar, where you can enter a reference record number in the Objs box or pick an object to specify a reference record number. (For more information on object referencing, see chapter 4, Understanding Terramodel Objects, “Referencing Objects To Other Objects”.)</p>
Type	<p>Adds to the selection set the specified object type (such as point, sets, or plines).</p> <p>When the focus is on the select control, you can select an object on the screen and Terramodel will add to the selection set all objects that are of the type you selected. Or, click the button next to the control to open the <i>Select by Type</i> dialog box.</p>
View	<p>Adds to the selection set all objects within a specified view. Specify a view by picking an object (turned on in Search) within the view you want to select, or click the button next to the select control to open the <i>Select by View</i> dialog box.</p>
Window	<p>Adds all objects within specified window corners to the selection set.</p> <p>To designate a window, pick the corners directly from the graphics screen, or click the button next to the select control to open the Window command bar, where you can enter north and east (or X and Y) coordinates for the window corners in the From and To boxes. Terramodel will not add objects that are turned off on your graphics screen or objects turned off in Search. Only visible objects can be selected.</p>
Expression	<p>Select the Expression option by clicking your right mouse button to display the Select options and picking Expression. For details, see “Select Expressions”.</p>

Select by Color Dialog Box

To add to the selection set objects that have a specified color, choose Color as the selection option. You may type in the color number or pick it from the *Select by Color* dialog box. Press the space bar or click the button next to the select control to open the *Select by Color* dialog box.

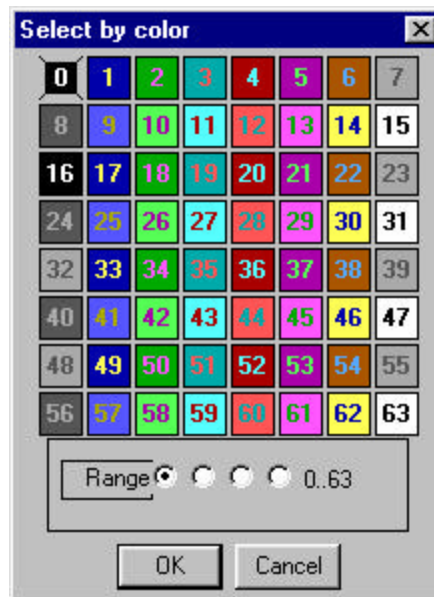


Figure 6-3 *Select by Color* Dialog Box

When the *Select by Color* dialog box is not open you can also select a color by clicking an object in your file that has the color you want to use.

Select by Layer(s) Dialog Box

This select option adds to the selection set objects that are on a specified layer. Press the space bar or click the button next to the select control to open the *Select by Layer(s)* dialog box, where you can pick one or more layers from the list of layers in your project.



Figure 6-4 Select by Layer Dialog Box

To pick one layer for your selection set, highlight it with your mouse. To pick several sequential layers, hold down your left mouse button and drag the cursor down the list of layers. To pick several non-sequential layers, hold down the CTRL button while picking them with your mouse. To select all layers use “CTRL/”.

Select by Linetype Dialog Box

The Linetype select option adds to the selection set objects (in the same view) that have a specified linetype. Press the space bar or click the button next to the select control to open the *Select by Linetype* dialog box where you can pick one or more linetypes from the list of linetypes in your project.

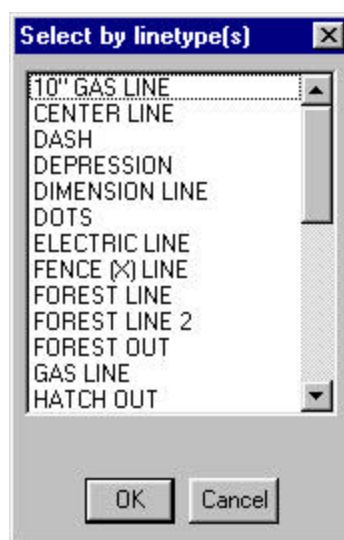


Figure 6-5 Select by Linetype Dialog Box

To pick one linetype for your selection set, highlight it with your mouse. To pick several sequential linetypes, hold down your left mouse button and drag the cursor down the list of linetypes. To pick several non-sequential linetypes, hold down the CTRL button while picking them with your mouse. To select all layers use “CTRL/”.

Select by Name Dialog Box

The Name select option adds to the selection set objects (in the same view) that have a specified name. Press the space bar or click the button next to the select control to open the *Select by Name* dialog box, where you can enter a name in the Mask box.

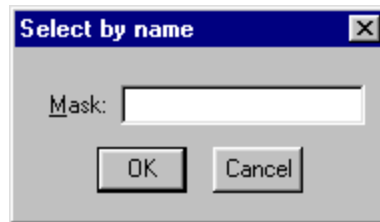


Figure 6-6 Select by Name Dialog Box

The object name can have wildcard characters so that you can select several names. When selecting by name the name used is case sensitive.

You can also specify a name by clicking an object in your file that has the name you want to use.

NearLine Command Bar

The NearLine select option adds to the selection set objects that lie within a specified distance of a selected line. Press the space bar or click the button next to the select control to open the NearLine command bar, where you can enter an offset and pick a line. For offset distances of zero, if the focus is on the control you can select a line without opening the NearLine command bar.

For a set line, **Terramodel** includes the points that define the set in the selection set. If the offset distance is given as zero, the selection set will consist of only the points that define the set. The offset distance must be greater than zero to select points that fall exactly on a polyline.

It is not necessary for polylines or sets to be turned on in **Search** to be picked as the selection set line.

Select by Point Number Dialog Box

The Points select option adds to the selection set a point having a specified number, or points within a point number range. Press the space bar or click the button next to the select control to open the *Select by Point Number* dialog box, where you enter a number or range of numbers in the Points box.

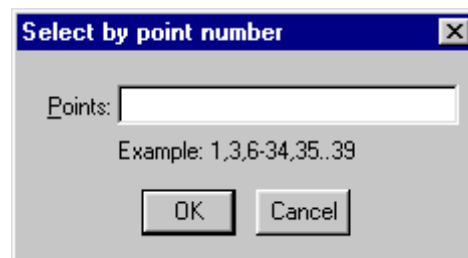


Figure 6-7 *Select by Point Number Dialog Box*

Enter multiple point numbers by separating them with commas. Specify a point number range by typing the first point number and last point number, separated by a hyphen or two periods (example: 1-100, or 1..100). This option does not support point ids (alphanumeric point numbers).

Select by Type Dialog Box

The Type select option adds to the selection set the specified object type (such as point, sets, or plines). Press the space bar or click the button next to the select control to open the *Select by Type* dialog box, where you can check the boxes of the object types you want to add to the selection set.

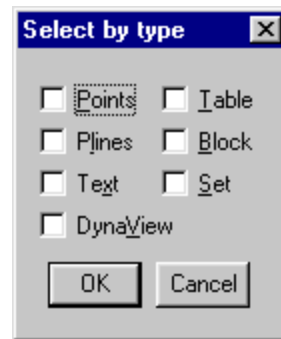


Figure 6-8 Select by Type Dialog Box

Select by View Dialog Box

The View select option adds to the selection set all objects within a specified view. Press the space bar or click the button next to the select control to open the *Select by View* dialog box, where you can check the boxes of the views whose objects you want to add to the selection set. The current view will be automatically checked.

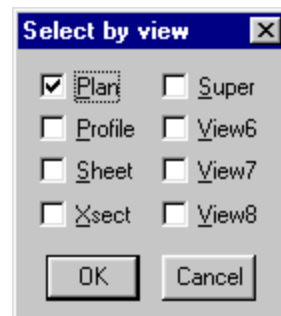


Figure 6-9 Select by View Dialog Box

Select By Window Command Bar

This select option adds all objects within specified window corners to the selection set. Press the space bar or click the button next to the select control to open the *window* command bar, where you can specify the corners of the window.

Figure 6-10 Select by Window Command Bar

Select Expressions

As you add selections using the select options, **Terramodel** builds a selection expression that it evaluates once the selection set is closed. You can view this expression by opening the *Select by Expression* dialog box. (You can select the Expression option only by clicking your right mouse button to display the Select options and picking Expression.) If necessary, you can edit any of the expressions from within this dialog box. Each expression entry must be separated by the expression operator, and include a select option (color, name, window, etc.) whose parameters are enclosed by parentheses (). This option is provided primarily for informational purposes and it is not intended as a typical way for the average user to specify a selection.



The first parameter in each entry represents the search status (object types that can be selected) for that entry. This code is assigned by Terramodel and should not be changed. Where applicable, a second parameter identifies the view.

Each select option type is accompanied by a specific type of parameter, enclosed in parentheses. Table 6-3 lists each type of parameter and an example of how it is written.

Table 6-3 Select Expressions

Select Options & Parameters	Example
Color(*ss,view ID,color number)	Color(0x6bb,1,4)
Crossing(*ss,view ID,N ₁ ,E ₁ ,N ₂ ,E ₂)	Crossing(0x6bb,2,50,60,2000,100)

Select Options & Parameters	Example
Elevation(*ss,view ID,beg elev, end elev)	Elev (0x6bb,1,1025.24,1030)
Group(*ss,view ID,group code number)	Group (0x6bb,2,2030)
Inside(*ss,boundary record #)	Inside (0x6bb,35)
Layer(*ss,view ID,layer1{ layer2} { layer3})	Layer (0x6bb,2,CONTOURS LOTS)
Linetype(*ss,view ID, linetype name)	Linetype (0x6bb,2,SHORT DASH)
Name(ss,view ID,object name)	Name (0x6bb,1,MANHOLE)
NearLine(ss,offset distance, line record number)	NearLine (0x6bb,-25,103)
Offline(ss,line record #)	Offline (0x6bb,103)
Points(p1{,p2}{,p ⁿ >p ^{n+x})	Points (5,13,120 >125)
Record(obj. record #)	Record (103)
Reference(ss,view ID, reference record #)	Reference (0x6bb,1,324)
Type(view ID,object type)	Type (1,PLINE)
View(ss,view name)	View (0x6bb,Plan)
Window(ss,view ID,N ₁ ,E ₁ ,N ₂ ,E ₂)	Window (0x6bb,2,5000,4500,5050,4600)

*ss = search status

Using Wildcard Characters Within Names

Whenever **Terramodel** requests a name, you can use wildcard characters. Wildcard characters let you select multiple objects that have different names, but have certain common characters in their names. Wildcards can be used in expressions for select options like Name, Layer, and Linetype. By using the wildcard characters in Table 6-4 in your expressions, you can increase or decrease your range of selected objects.

Table 6-4 Wildcard Characters

Wildcard Characters	Description	This example...	Returns...
# (Pound)	Matches any numeric digit.	Lotline#	Lotline3, Lotline50
@ (At)	Matches any alpha character.	Lotline@	LotlineA, Lotlineb
. (Period)	Matches any non-alphanumeric character.	Lotline.	Lotline+, Lotline/
* (Asterisk)	Matches any string, including the null string. It can be used anywhere in the search pattern: at the beginning, middle or end of the string.	*CON*	Contour, Index Contour
? (Question mark)	Matches any single character.	DES?PTS	DES 2 PTS, DES N PTS
~ (Tilde)	Matches anything but the pattern.	~Lotline	Any pattern other than Lotline.
[...]	Matches any of the characters enclosed in the brackets.	[AB]C	AC or BC

Wildcard Characters	Description	This example...	Returns...
[~...]	Matches any character <i>not</i> enclosed in the brackets.	[~con]*	Anything <i>not</i> beginning with c,o, or n
- (Hyphen)	Used inside brackets to specify a range for a single character	[A-J]C	AC, BC, CC JC
‘ (Reverse single quote)	Reads literally any special character immediately following the wildcard	*con	*con

Expression Operators

PREVIOUS

This option adds the objects from the previous selection set.

= (EQUALS)

This option re-initializes the selection set with the next pick. Use this option when you want to abandon the current selection expression but not exit the command.

+ (OR)

This option adds to the current selection set.

Example:

```
Layer (points) +Layer (contours)
```

Selects all objects on both layers in all views.

– (AND NOT)

This option removes from the current selection set.

Example:

```
Layer (*con*) - Layer (contours)
```

Selects all objects on layers with the letters “con” in their layer names, excluding the layer “contours”.

^ (XOR)

This option specifies condition A or B, but both conditions do not have to be met for the objects to be selected.

Example:

```
Color (12) ^ Layer (LOTS)
```

Selects either objects having a color of 12 or objects on layer LOTS but not objects that have color 12 and are on layer LOTS.

*** (AND)**

This option dictates that the selected objects must meet conditions A and B.

Example:

```
Color (12) * Layer (LOTS)
```

Selects only objects that are on Layer LOTS *and* have a color of 12.

! (NOT)

This option excludes the specified condition.

Example:

```
Layer (1) ! Color (12)
```

Selects all objects on layer 1 except for those that have a color of 12.

Expression operators can be nested using parenthesis as desired.

Searching for Objects

Often when you are manipulating objects in your project, you want to be able to select only certain kinds of objects. **Search** controls your ability to manipulate specific types of objects when you use certain commands. If you type **Search** at the command bar, select the **Search Settings** option from the **SETTINGS** menu, or enter CTRL+S, **Terramodel** opens the *Search Settings* dialog box.

Search Settings Dialog Box

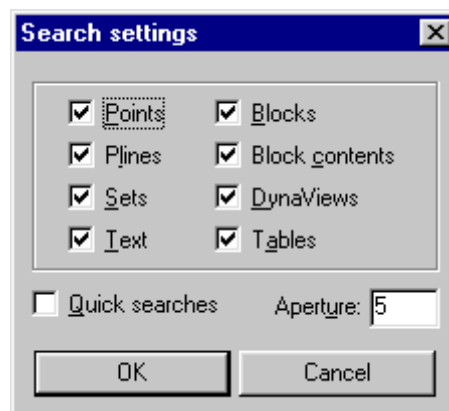


Figure 6-11 Search Settings Dialog Box

- **Points, Plines, etc.**—When you check or clear the check box beside an object type, you toggle **Terramodel**'s ability to “see” these objects for the purpose of manipulating them. The more object types that are turned on under **Search**, the longer it takes **Terramodel** to search for specific items. For this reason, you should be aware of what is turned on under **Search** at all times and feel comfortable in changing the search status of objects often.
- **Quick searches**—Forces **Terramodel** to select the first object it finds inside the cursor box. When **Quick search** is disabled, **Terramodel** selects the object that is closest to the center of the cursor box.
- **Aperture**—Sets the size (measured in pixels) of your screen pick box. Changing the aperture gives you added control over which objects are found. This is the same setting as in the **System** command.

Set labels and the segment labels created when labeling tables are not objects but are attributes and can not be directly selected. Commands that require that you pick a certain type of object ignore the select status and allow you to pick only the object type required. If a command allows you to pick a set or polyline the search status is used for sets and plines unless both are turned off then both are searched for. You can only snap to or list objects whose search status is on. You can only snap to objects that are on layers that have their snap status on.

Object Snap Modes

Object snap modes increase **Terramodel**'s flexibility in creating objects and constructing lines. You can use snap modes to create a point at a station and offset from an alignment using **Point**, or to determine an elevation on a DTM surface using **Spot**. Snap modes also make **Terramodel** easier to use by providing identical options for almost all commands.

You can combine snap modes within a single command. For example, you can locate several points, and snap to a line to use its bearing to draw a polyline, all in a single command. Not only does this save time, but it reduces the number of possible errors.

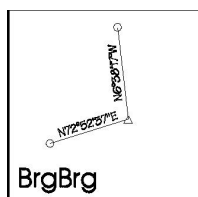
Object Snap modes are available during a command when an indicator such as the word **Point** or **Bearing** appears in the right corner of the Status bar. The snap modes include **Point Snap**, **Bearing Snap**, **Distance Snap**, **Vertical Snap**, and **Angle Snap**, and are discussed in this section. The options contained in the **Point Snap Menu** are shown in Figure 6-12.

Point Snap Modes

To display the Point Snap modes, right-click the mouse anywhere within the view window whenever *Point* appears in the right box of the Status bar.

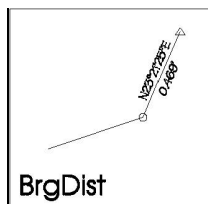
BrgBrg	Ins	OffInt
BrgDist	Intersect	OffIntHal
Center	IntHal	Perp
DxDy	Mid	PerpHal
End	Mid2Pt	PI
Factor	Name	Point
FactHal	Nearest	StaElev
Fact2Pt	Offset beg	Quadrant
Free	Offset end	Tangent
Gravity	OffHal	XY

Figure 6-12 Point Snap Modes



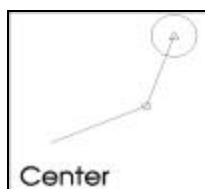
BrgBrg (Bearing-Bearing)

Snaps to the intersection of two lines defined by a point and bearing and a second point and bearing. No elevation is set using this option.



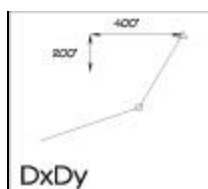
BrgDist (Bearing-Distance)

Snaps to a point based on a beginning point, bearing and distance. No elevation is set using this option.



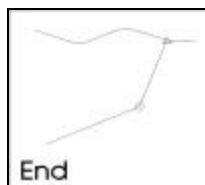
Center

Snaps to the center point of an arc, parabolic curve segment or spiral curve segment by selecting the arc. The elevation of the center point will be used as the Z value.



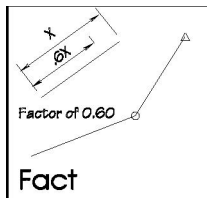
DxDy (Relative Distance)

Snaps to a point defined by a relative “x” and “y” distance from an origin point. The origin point defaults to the last point to which the rubber band was attached. The elevation of the origin point will be used as the Z value.



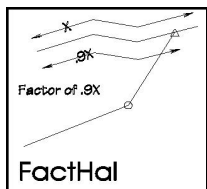
End

Snaps to the end of a line or arc segment. **Terramodel** will snap to the end of the line that is closest to where the line was picked. The elevation of the end point will be used as the Z value.



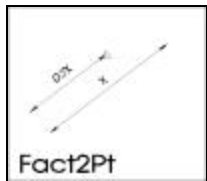
Factor

Snaps to a factored distance along a selected line. Valid factor entries are between 0.0 and 1.0, where 1.0 causes **Terramodel** to snap to the end of the line closest to where it was picked. If the line segment has an elevation the Z value will be interpolated from the segment.



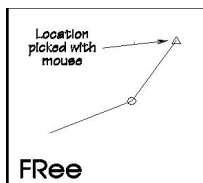
FactHal

Snaps to a line at a factored distance from the end of the line (based on the entire line rather than a line segment). The computed distance will originate from the end with a station of zero. If the line segment has an elevation the Z value will be interpolated from the segment.



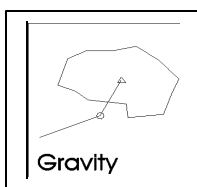
Fact2Pt

Snaps to a factored distance between two points. The points do not have to be existing point objects in the project file. **Terramodel** will measure the factored distance from the first point location that you specify. If the two points have an elevation the Z value will be interpolated between the two points.



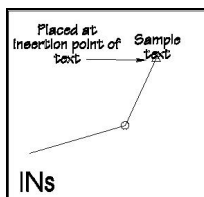
Free

Snap to a point using only the cursor position. No elevation is set using this option.



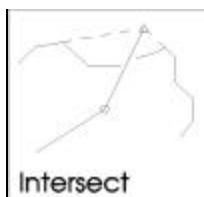
Gravity

Snaps to the center of gravity of a selected object (set or polyline). This option is useful for locating the center of a plotbox, as well as locating the center of gravity of complex shapes. If the line segment picked to select the figure has an elevation the Z value will be interpolated from the segment.



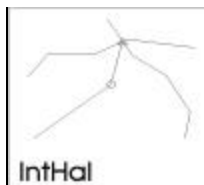
Ins (Insertion Point)

Snaps to the insertion point of a table, block, text or dynaview object. The elevation will be used from the object selected.



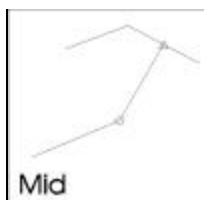
Intersect

Snaps to the projected intersection of two line (or arc) segments. If there are several projected intersections of the two selected lines, **Terramodel** uses the intersection closest to where the lines were picked. No elevation is set using this option.



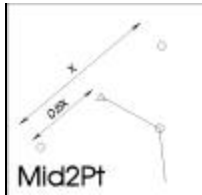
IntHal

Snaps to the projected intersection of two HALs. Where there are multiple intersections, **Terramodel** uses the intersection closest to where the lines were picked. No elevation is set using this option.



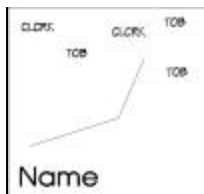
Mid

Snaps to the middle of a line (or arc) segment. If the line segment has an elevation the Z value will be interpolated from the segment.



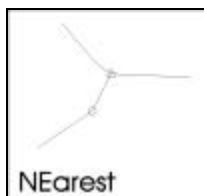
Mid2Pt

Snaps to a location midway between two points. The points do not have to be existing point objects in the project file. If the two points have an elevation the Z value will be interpolated from the two points.



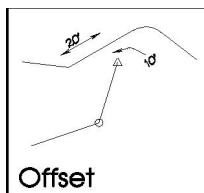
Name

Snaps to a selected point by selects a point from all points that have the same name. The cursor rubber band attaches to the lowest (in point number) of the selected points and displays the point number. You can press ENTER to advance the cursor through the points in numerical order, or select the point attached to the rubber band with your cursor. If you advance through the entire selection of points without picking one, Terramodel displays “Point not found” and returns to the Point prompt. If the point has an elevation it will be used as the Z value.



Nearest

Snaps to a line and uses the coordinates on the line nearest to the cursor location. If the line segment has an elevation the Z value will be interpolated from the segment.

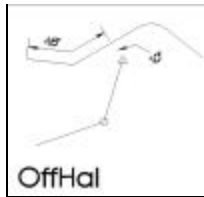


Offset Beg/End

Snaps to an offset from a selected line or arc segment. After you select a segment, you enter a distance from the beginning of the segment and a negative or positive offset distance from the line. Use *offset beg* to indicate that end of the segment closet to the point where the line was selected is the beginning of the line, the station is measured from this point. Use *offset end* to indicate that end of the segment closet to the point where the line was selected is the end of the line. The station will be measured from the beginning (the other end) of the line. Negative offset values are to the left of the line, looking in the direction of increasing distances. If the line segment has an elevation the Z value will be interpolated from the segment at the station specified.

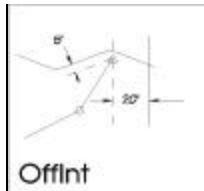


You can use this option to select a specific position along an arc by making the offset distance zero.



OffHal

Functions identically to Offset, except **Terramodel** computes the distance along the entire length of the line instead of the segment. The distance you enter is based on the stationing of the line. If the line segment has an elevation the Z value will be interpolated from the segment at the station specified.



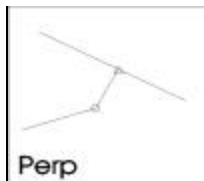
OffInt

Snaps to the intersection of offsets from two selected line or arc segments. For each selected segment, you enter a negative or positive offset distance from the line. **Terramodel** snaps to the intersection of the offsets you entered. Negative offset values are to the left of the line, looking in the direction of increasing stations. No elevation is set using this option.



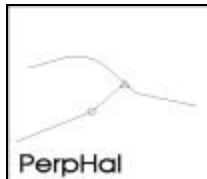
OffIntHal

Functions identically to OffInt, but **Terramodel** projects the offsets from the entire lines rather than line or arc segments. No elevation is set using this option.



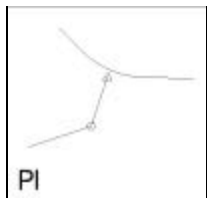
Perp

Snaps to an intersection of a perpendicular from the previous point to a selected line or arc segment. If the line segment has an elevation the Z value will be interpolated from the segment.



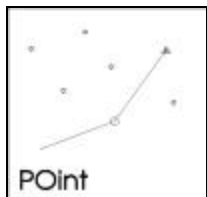
PerpHal

Functions identically to Perp except **Terramodel** uses the entire line or arc for its computations. If the line segment has an elevation the Z value will be interpolated from the segment.



PI

Snaps to the PI (point of intersection) of a selected arc, spiral, or parabola. No elevation is set using this option.



Point

Snaps to an existing point object. If the point has an elevation it will be used as the Z value.

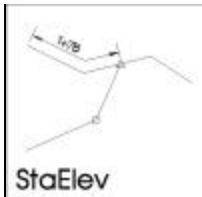


Quadrant

Snaps to the 0-, 90-, 180-, or 270-degree quadrants on a selected arc. **Terramodel** uses the quadrant that is closest to where the arc is picked. No elevation is set using this option.

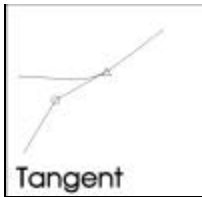


The Quadrant option is not valid with spiral or parabolic curves.



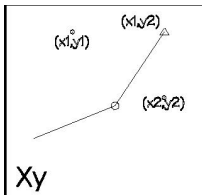
StaElev

This option is only valid in the profile view node. Snaps to a point on a polyline given the polyline and the desired station.



Tangent

Snaps to a selected arc at the tangent from the previous point. No elevation is set using this option.



XY

Snaps to the X-coordinate of the first selected point and the Y-coordinate of the second selected point. If the Y point has an elevation it will be used for the Z value.

Bearing Snap Modes

To display the Bearing Snap modes, right-click the mouse anywhere within the view window whenever *Bearing* appears in the right box of the Status bar.

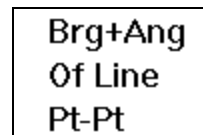
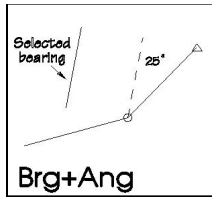
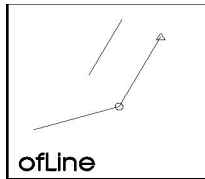


Figure 6-13 Bearing Snap Modes



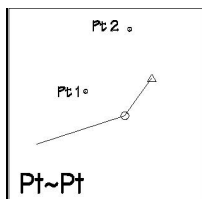
Brg+Ang

Specifies a new bearing by adding an angle to a given bearing.



Of Line

Uses the bearing of a given line. Which end of the line you pick determines the direction of the bearing.



Pt-Pt

Defines a bearing between any two given points. The direction of the bearing is from the first point to the second point. For each of the points specified the point snap modes made be used.

Distance Snap Modes

To display the Distance Snap modes, right-click the mouse anywhere within the view window whenever *Distance* appears in the right box of the Status bar.

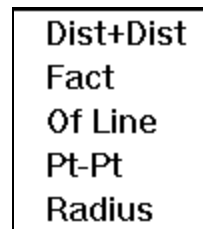
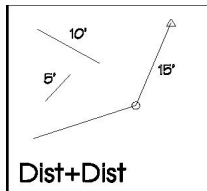


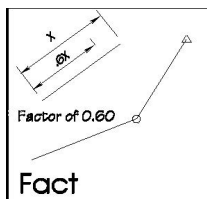
Figure 6-14 Distance Snap Modes



Dist+Dist

Dist + Dist

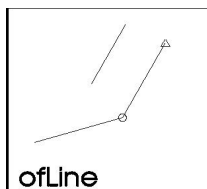
Computes a distance by adding two distances. Either distance can be negative.



Fact

Fact

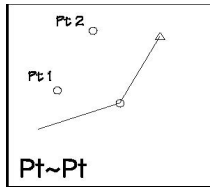
Computes a distance using a specified factor applied to a given line segment.



ofLine

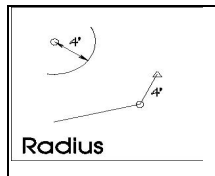
Of Line

Computes a distance value from the length of a selected segment.



Pt-Pt

Computes the distance between any two given points.



Radius

Uses the radius distance of a selected arc.

Vertical Snap Modes

To display the Vertical Snap modes, right-click the mouse anywhere within the view window whenever *Vertical* appears in the right box of the Status bar.

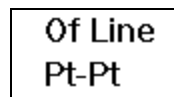
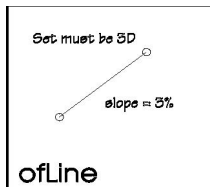
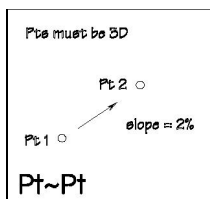


Figure 6-15 Vertical Snap Modes



Of Line

Uses the vertical angle of a selected set segment. The set you select must contain points with elevations. A line may also be selected from the profile view to specify the angle.



Pt-Pt

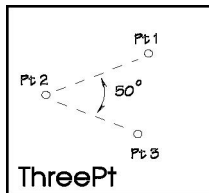
Lets you define a vertical angle between any two given points. The points you specified must have elevations. Points may also be specified in the profile view to specify the angle. The point snap modes may be used to specify each of the points.

Angle Snap Modes

To display the Angle Snap modes, right-click the mouse anywhere within the view window whenever Angle appears in the right box of the Status bar.

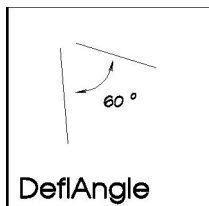
ThreePt
DeflAngle

Figure 6-16 Angle Snap Modes



ThreePt

Computes an angle from three given points. The point control is used to specify each point.



DeflAngle

Computes an angle from the bearings of two given lines. The bearing control is used to specify each bearing.

Elevation Snap Modes

To display the Elevation Snap modes, right-click the mouse anywhere within the view window whenever Elevation appears in the right box of the Status bar.

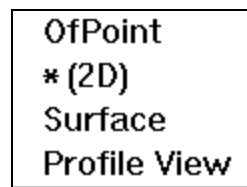


Figure 6-17 Elevation Snap Modes

OfPoint

Opens a command bar with a point locate control. The elevation of the point specified will be used.

*(2D)

Sets the elevation to the value used to indicate a 2D point.

Surface

Opens a command bar prompting for a DTM layer and point on the DTM layer. The elevation of the X,Y position on the surface will be used as the elevation.

Profile View

Opens a command bar asking for a point in the profile view. The elevation of the point selected will be used. The Station Elevation point snap mode can then be used to select an alignment and station on the alignment to interpolate an elevation.

Station Snap Modes

To display the Station Snap modes, right-click the mouse anywhere within the view window whenever Station appears in the right box of the Status bar.

AtPoint

Opens a point command bar with a point control so that an exact point can be used to indicate the station of the point on the alignment.

FromHAL

Opens a GetStation command bar so that the station of a point relative to a selected HAL can be specified. This option is primarily for views other than the plan view. This option is typically used when specifying a location in the profile view and the *Station Elevation* point snap mode is used. This allows you to specify a point along a line in the profile view at the station of a point relative to a HAL.

Offset Snap Modes

To display the Offset Snap modes, right-click the mouse anywhere within the view window whenever Offset appears in the right box of the Status bar.

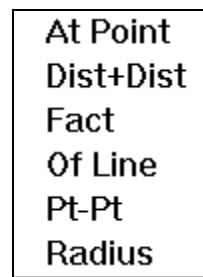


Figure 6-18 Offset Snap Modes

AtPoint

Opens a point command bar with a point control so that an exact point can be used to indicate the offset of the point along the alignment.

Dist+Dist

Computes a distance by adding two distances. Either distance can be negative.

Fact

Computes a distance using a specified factor applied to a given line segment.

Of Line

Computes a distance value from the length of a selected segment.

Pt-Pt

Computes the distance between any two given points.

Radius

Uses the radius distance of a selected arc.

Running Snap Modes

The running snap modes let you turn on or off certain point snap modes, and establish the order of precedence for the snap modes that are on. They are controlled via the *Running Snap Modes* dialog box.

Running Snap Modes Dialog Box

Open the *Running Snap Modes* dialog box by selecting **Running Snaps** from the **SETTINGS** menu, typing **Objsnap** at the command prompt, or pressing CTRL + O.

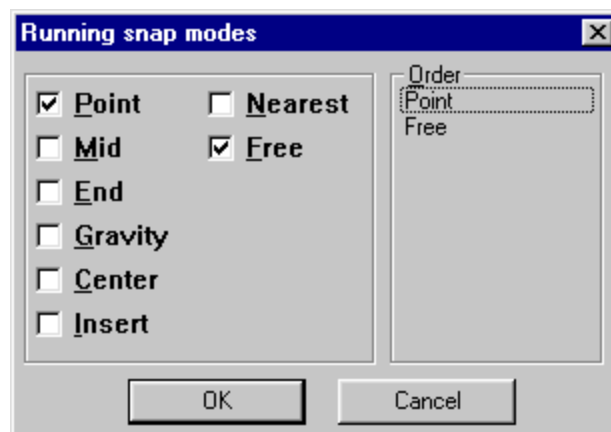


Figure 6-19 Running Snap Modes Dialog Box

By checking the box adjacent to a point snap mode, you can enable or disable it. When you turn on a snap modes, it appears in the Order list to the right. Set the order of precedence for your enabled snap modes by dragging them to different positions in the Order box. The top position has the highest precedence, the second position the second highest, and so on. You can change the snap status and precedence transparently as often as necessary while you are answering different prompts from within a single command.

For most situations, you should keep the Free snap mode enabled and it should hold the lowest priority. If it isn't checked and none of the other running snap modes are enabled, Terramodel can't pick during commands such as **Zoom**.

Chapter Seven

Creating a Digital Terrain Model

What Is a DTM?

A DTM (Digital Terrain Model) is a computer-generated, 3-dimensional model of a surface created from points that have x, y and z coordinates. In **Terramodel** a DTM is used as the basis for many advanced features such as calculating volumes and interpolating cross sections.

Three points (a triangle) are the minimum number of points required to uniquely define a plane. One way to define a surface is by a series of planes or triangles. The type of DTM that **Terramodel** uses is a Triangulated Irregular Network or TIN, which is simply a series of planes or triangles. Each triangle is defined by three points where each point knows which other points it is connected to. In **Terramodel** the process of determining the best triangles from the given points is called linking and the imaginary lines between points defining the triangles are called link lines. **Terramodel** uses the edges of these triangles (the link lines) to interpolate the locations of contours.

All engineers and surveyors are familiar with the complexity of actual land terrain—there are mountains, hills, valleys, and bodies of water that must be accounted for in a model. To control a model, we must have the ability to adjust not only the points but the links that connect the points. The topics in this chapter explain how you can control the link lines of the DTM.

Links and a DTM

The links between points are not true **Terramodel** objects, but are actually a property of the points on the DTM layer that provide **Terramodel** with a way to interpolate elevations and calculate volumes.

Links change dynamically as you make changes to your DTM. They control the direction of slope in the plane surfaces of the triangles. **Terramodel** uses several linking parameters that you should set prior to linking a DTM, since these settings are crucial to the accuracy of your model. The **LinkSet** command opens the *Link Settings* dialog box where you establish and change these link settings.

Link Settings Dialog Box

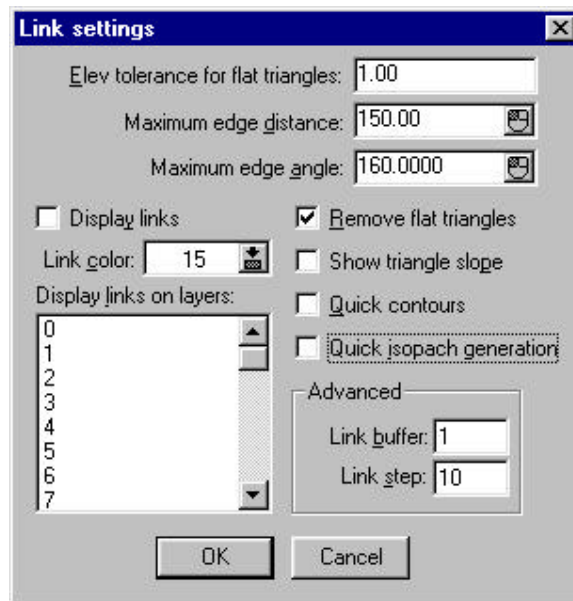


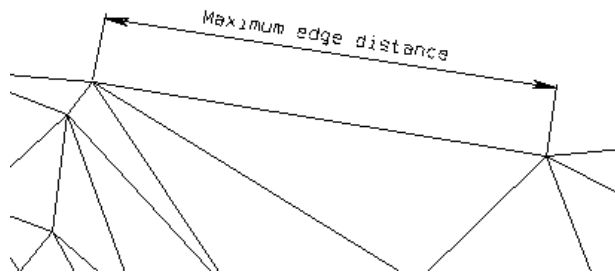
Figure 7-1 Link Settings Dialog Box

Elev Tolerance for Flat Triangles/Remove Flat Triangles

On some occasions you may be forced to trace in contour lines that will be used to create a DTM. In this event it is desirable to have the DTM closely emulate the original contour lines. The normal method of forming triangles tries to make triangles with equal interior angles. In areas where the contour lines form a “finger”, flat triangles can be created. You can force **Terramodel** to remove these flat triangles, when possible, and force the DTM to slope into these flat areas. This is done by connecting the points with a set line and telling **Terramodel** to remove flat triangles. **Terramodel** will check the elevation of the first point on a set against the remaining points on the set. If the difference in elevation is less than the tolerance value you set in Elev Tolerance for Flat Triangles, and you have checked the Remove Flat Triangles check box, **Terramodel** will modify the links to remove the flat triangles and create triangles that connect points that are not a part of the line. This process improves the modeling and contouring of this type of DTM, and we recommend it for this type of data.

We suggest a value of one-half of your contour interval as a good elevation tolerance. Whenever you change this value, you will have to re-link to update your surface.

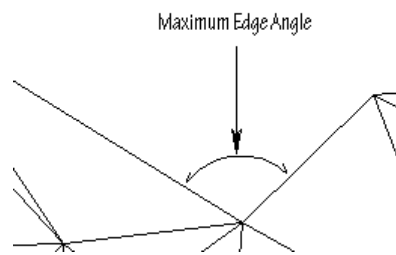
Maximum Edge Distance



This distance is the maximum distance that **Terramodel** will allow between two points that it links on the exterior edge of the DTM. If two points are farther apart than this distance, they will not be connected with a link line. This linking parameter is important because it prevents **Terramodel** from contouring and including in volumes calculations any areas that you really want to omit.

To determine what the maximum edge distance should be for your particular project, measure the distance along the DTM perimeter between points that have the farthest distance between them. Set the edge distance slightly less than that value, then link and contour the DTM using **Contour**. Turn on the links for that layer (see “Display Links” on page 266) and decide if the links and contours are satisfactory. If not, increase the link distance and repeat the process.

Maximum Edge Angle



This angle represents the maximum internal angle that **Terramodel** will allow between any three points along the edge of a DTM. If the internal angle is greater than this value, **Terramodel** will not connect the points opposite this angle with a link line. Setting this parameter also prevents **Terramodel** from contouring and calculating volumes for undesirable areas.

To set the maximum edge angle for your project, measure the internal angle between the two points that are farthest apart. Set the maximum angle at slightly less than that value, then link and contour the DTM. Turn on the links for the layer (see “Display Links”, below) and decide if the links and contours are satisfactory. If not, increase the maximum edge angle and repeat the process.

Display Links/Link Color/Display Links on Layers

To be able to judge if a DTM is being modeled correctly, it is necessary that you view the links. The Display Links check box toggles the visibility of links, the Link Color control lets you pick the color in which you can view the links, and the Display Links on Layers list box lets you pick the layers whose links you want to display. If you want to show the links for more than one layer, hold down the CTRL button while you click the layer selections. If the link color is set to 0, **Terramodel** will display the links in the colors of the layers on which the points reside. Consider using subtle colors to display the links, since bright colors tend to be overwhelming for showing a large number of links.



You can only display links for layers that have 3-D points on them.

Show Triangle Slope

When this box is checked, **Terramodel** will display the slope arrows for the DTM layer(s) highlighted in the Display links on Layer list box. **Terramodel** will display the slope arrows in the color selected in the Link Color control. If you select 0 for the slope arrow color, the arrows will be displayed in the color of their respective layers. These arrows are only displayed, and are not composed of polyline objects that you can modify.



You can display slope arrows only for layers that have 3-D points on them.

Quick Contours

When this box is checked, **Terramodel** will automatically interpolate and display straight-line contours (as if View Resolution Segments per Spline = 0 in **DisplaySet**) for the DTM layer(s) highlighted in the Display Links on Layer list box. **Terramodel** will display the contours in the color selected in the Link Color control. If you select 0 for the contour color, the contours will be displayed in the color of their respective layers. These contours are only displayed, and are not composed of polyline objects that you can modify.



You can display quick contours only for layers that have 3-D points on them.

Quick Isopach Generation

This option is used by the volume command to determine how the isopach is generated. For the most accurate results *do not* enable this option.

Surface-to-surface volumes are calculated by first computing the isopach between two DTMs (Digital Terrain Models) or between a datum and a DTM. This is done by projecting each point onto the other surface, interpolating the corresponding elevation, and then storing the elevation difference as the elevation of the generated isopach point. The breaklines (SETs) of both surfaces are used to generate isopach breaklines in a similar manner. Additional points are inserted into the isopach breaklines where they cross DTM triangles. If the *Quick Isopach* option is enabled, the isopach is then linked and volumes are calculated. For many projects stopping the calculations at this point will produce an isopach and volumes that are accurate to within the user's needs and the methodology of recording elevations and creating a DTM. This option has the added benefit of reducing the number of calculations and duplicate points that will be created (i.e. you get done faster).

If the *Quick Isopach* option is not enabled, additional computations will be performed that are intended to assure that the relative shapes of the DTMs are completely held in the isopach. Next all link lines are "walked" and where they cross between the two surfaces points are created. This isopach is linked. All of the links of the two DTM's are walked and a test is made to see if there are link lines between the points on the isopach. If a link line is not found a set line is placed between the points. When complete volumes are calculated from the resultant isopach.

Advanced

Link Buffer—The link buffer is the additional memory **Terramodel** allocates for the links connected to each point. Allocating more memory lets **Terramodel** function more efficiently, but the level of efficiency depends on the size of your project files and your computer's memory. A value of 0 is memory-efficient, but slows down the linking process. A value of 10 increases the speed of linking, but wastes memory. We recommend using a link buffer value of 2.

Link Step—The link step lets you establish the increment **Terramodel** will use in adding points from a points file to the surface. For instance, if you set a value of 10, **Terramodel** will add every 10th point to the surface as it makes its initial pass through the points. It then starts at the beginning of the file and adds the 2nd point, then the 12th point, and so on. For a step value of 10, **Terramodel** will make 10 passes through the database during the process of adding points. This procedure allows **Terramodel** to add points in a random fashion, forming fewer links and thereby speeding up the linking process. A step value of 1 adds points in the order in which they are read into the database, potentially creating more links per point than are allowable (a point can have up to 300 links attached to it) and slowing down the process dramatically. We recommend a link step value of 10.

Changing the Links with DtmEdge

In addition to the Maximum Edge Distance and Maximum Edge Angle link parameters, you can also use **DtmEdge** to control the exterior links.

When all DTM points are enclosed by a set or series of sets, no links can form outside of the set lines. **DtmEdge** creates a closed set that connects all of the points along the perimeter of a DTM. After **DtmEdge** creates the set, you can break or modify it as necessary.

Adding Breaklines to a DTM

When a terrain model contains an irregular surface such as a ditch, ridge, railway or curb, you want to be able to prevent points that are on opposite sides of the irregularity from being linked. This will ensure that the model accurately reflects the real land surface, and will make contouring, volume calculations, site design and other activities much more accurate.

Breaklines, also referred to as structural or discontinuity lines, let you revise how the triangle links should be created. In **Terramodel**, you create breaklines by using set objects to connect points along the top of a ridge, flow line of a ditch, top of curbs and gutters, or any feature that you want to prevent links from crossing. For sets to affect the linking process, they must:

1. Occupy the same DTM layer as the points that form the surface you want them to influence, and
2. Connect 3-D points.

Sets also have the ability to affect the appearance of contours that cross them. Contours that have been splined (smoothed) will not necessarily show sharp breaks in the terrain unless you remove the splining. You can apply a smoothness to sets with the **SetSmooth** command that will cause the crossing contours to look smooth, sharp or hard. The four types of smoothness are:

- **Dead**—Contours will not be drawn inside of a closed set boundary line. This smoothness is used where contours inside of building footprints would be undesirable. The entire line is generated and then clipped inside the closed dead area. The DTM continues through all dead areas.
- **Soft**—Contours illustrate relatively smooth transitions, such as in natural features. A single vertex is created as the contour line crosses the set line.
- **Hard**—Contours bend more sharply around set lines. A hard break in the contour is created by inserting two vertices in the polyline on either side of the intersection with the set line.

- **Sharp**—Contours depict abrupt changes in a polyline where it crosses a set line. This smoothness is used for such features as curbs and gutters or edges of concrete pads. A sharp set line is created by inserting three vertices in the polyline on either side of and at its intersection with the set line.

With a smoothness of Dead, the DTM surface still extends inside of the dead areas, although the contours are clipped to the boundaries.

To show how you can use sets to model a surface, assume that a creek flows through three data points (points 3, 4, and 5 in Figure 7-2) and all other points are at a higher elevation along the sides of the creek. Note that without any breaklines to control them, triangle links join points 2 and 7, and points 1 and 6.

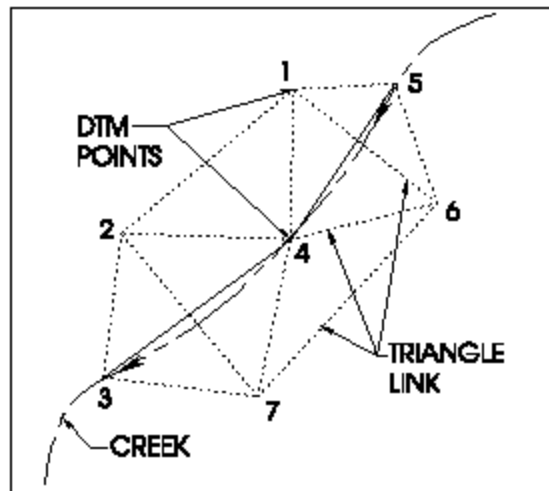


Figure 7-2 Example DTM Model Before Adding Breaklines

If you look at a cross section between points 1 and 6 (or 2 and 7), it might look like this:

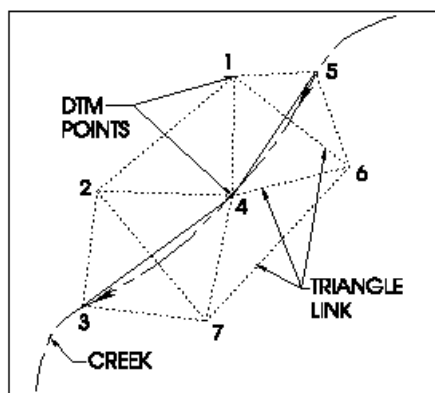


Figure 7-3 Cross Section of Example DTM Model Before Adding Breaklines

Since you want your TIN to represent the actual ground as closely as possible, you can revise the triangle links between points 1 and 6 and points 2 and 7 to run along the creek instead of at right angles to it. If you connect points 3, 4 and 5 with a set line and re-link the DTM, the links will look like this:

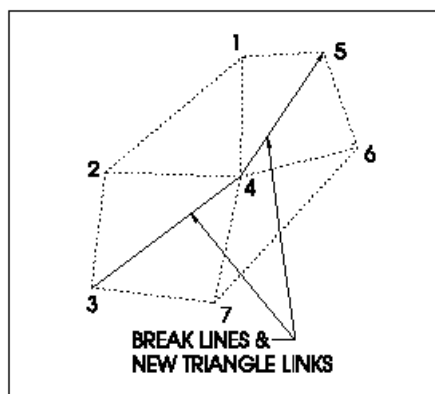


Figure 7-4 Example DTM Model After Adding Breaklines

Note that the links are now forced *along* the creek, and points 2 and 7, and 1 and 6 are no longer connected (links cannot cross set lines). Changing the links by adding breaklines more closely approximates the true ground surface, and consequently produces more accurate contours and volumes calculations.

Spline types

When a contour line is created from a DTM, the location of the line is determined by interpolating the elevation from the edges (links) of the triangles. A point (vertex) along the contour is created on the link line at the appropriate elevation. The basic contour line is composed of a series of straight segments with the segment ends always falling along the triangle edges. The line intersects the DTM along its entire length at the proper elevation.

Although it may be beneficial to be able to see all of the straight segments of our interpolated contour line, generally we expect to see contour lines as smooth, curved lines. **Terramodel** supports two types of “smooth” splined lines, the cubic b-spline and the Overhauser (also known as the Catmull-Rom) spline. The basic difference is that the b-splined lines have their curves “pulled” toward the vertex points, while the Overhauser spline lines have their curves passing through the vertex points.

As the DTM is composed of flat triangles any smoothing of contour lines misrepresents our mathematical model. The question then is, which method best creates a visually pleasing yet somewhat accurate representation. The option is ultimately left to the user, however historically contour lines are typically represented by the cubic b-spline.

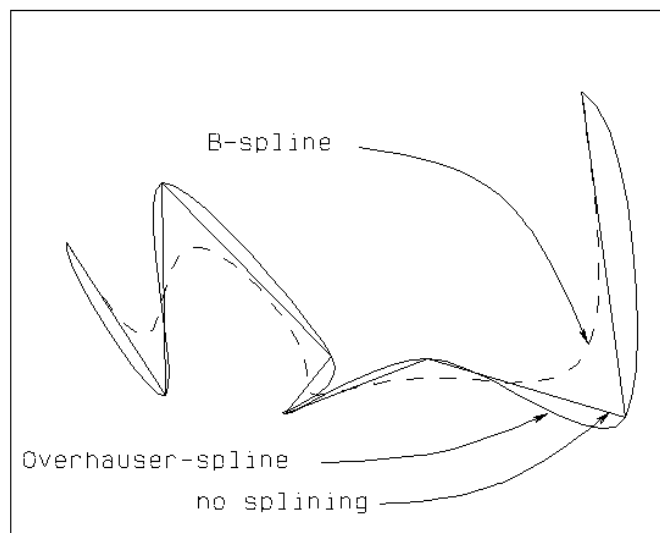


Figure 7-5 Spline types using the same spline frame

Controlling the DTM edge

When all of the points of a DTM layer are enclosed by one or more sets with straight line segments, on the same layer, connecting 3D points of the same layer, the link lines will not extend beyond the edge defined by the sets. If you need to control the edge of the DTM this rule can be applied. A quick way to create a set line around the current edge is to use the **DTMEdge** command. This set can then be broken and adjustments made. If the above rules are violated the edge will not be controlled. The following table will help you to find out how the rules are being violated if the edge is not being controlled. (See chapter 6, **Using Terramodel Custom Controls**, for instructions in the use of select options to create selection sets.)

Table 7-1 DTM Edge Requirements

Requirement	How to Confirm
All points contained in the DTM edge set must be 3-D.	List Points by Offline, selecting the sets that make up the DTM edge. Be sure all points have elevations.
All points and sets making up the DTM edge must be on the same layer.	List Objects by Offline, selecting the sets making up the DTM edge. Be sure they are all on the same DTM layer.
All points on the DTM layer that fall outside of the DTM must be 2-D.	Select points by elevation to see which are 2D and 3D.
No duplicate sets along the DTM edge.	List Sets by the DTM Layer, making sure all sets are accounted for.
No points along the DTM edge that are not included in the DTM edge set.	List Points by Offline, picking the DTM edge and giving an offset of 0.01. Remove from the selection set by Offline, picking the DTM edge again.

Multilayer DTMs

Terramodel can use multiple layers as a single DTM layer. This is accomplished by using the **MultiLayerDTM** command. This command enables a layer to be used as a base layer for a multi-layer DTM. When the base layer is specified as a DTM, the member layers are treated as a single DTM. The member layers are defined by creating a layer list with the same name as the base layer and including the desired layers as members of the layer list. The **MultiLayerDTM** command controls the setting for the specified layer and then opens the *Layer list setting* dialog box so that the appropriate layer list can be created.

A layer that is a member of one multi-layer DTM cannot be a member of another multi-layer, and cannot be used as a single layer DTM.

Chapter Eight

Getting Digital Data into/out of Terramodel

Introduction

Terramodel's import and export features allow you to get digital data into and out of a Terramodel project. Two-dimensional and three-dimensional points, raw survey data, surface data, roadway data and CAD data can be imported and exported. The import command includes downloading data from a wide array of survey devices and importing that data into Terramodel data objects. Import also includes importing data formatted by many software packages. Export includes exporting Terramodel data into specific software formats as well as exporting and uploading data to survey devices.

The download and import functions are linked within one command (**Import**), as are the export and upload functions (**Export**). For example, you can execute a single script to download raw survey data from a data collector, copy that raw data file on disk for archival purposes, and import the raw data into a Terramodel project file. Terramodel also offers a Raw Data Editor (see chapter 9) that performs the complex computations required to transform sets of raw survey data into a set of coordinates.

You can download/import or export/upload data using import or export script files. These scripts allow you to store each import or export process and to customize it exactly to your needs. Each script displays the dialog boxes required by the specific import or export process. To the extent this process is standardized, the script can be configured to automate the process.

Use the Import Script Manager (**ImportSMgr**) and Export Script Manager (**ExportSMgr**) to create, edit, and delete the import scripts provided at installation, or to create and manage your own scripts. The instructions in this chapter will aid you in creating a suite of custom scripts for the data you work with regularly.

Terramodel Download Options

Terramodel supports downloading from current Trimble devices and many older models.

TRIMBLE DEVICES

- TS315
- Trimble 3300
- Trimble 3600 (Elta, GDM, TDS)
- Trimble 5600 (Elta, GDM, TDS)
- Trimble Constructor 50/55
- Trimble DiNi (M5 format)
- Survey Controller
- Survey Data Card
- older devices including Geodimeter devices and Zeiss Elta (M5 format) devices

TDS DEVICES

- TDS Survey Pro CE
- older devices including TDS Husky and TDS running on an HP48

MISCELLANEOUS OTHER DEVICES AND INSTRUMENTS

- Leica GIF (a/n and gsi)
- Nikon DTM 310/500
- Pentax PCS 300
- Sokkia (SDR 20/33)
- SMI V6 or on HP 48 V5
- TPC on Husky

The following devices can be downloaded and imported by using the <Generic Instrument> device format and the appropriate logger file.

- Topcon GTS-3/700
- DGM
- Ellar
- Nikon AP700/Nik
- NSS
- Panterra
- Psion
- Steanne
- Survpak

Complete download/import/export/upload capability is not offered for every device listed.

The devices that Terramodel can communicate with is updated regularly, and the list above is necessarily incomplete. If you do not find your instrument or data file format in these lists, please call your local distributor to see if it has been added recently.

Import File Format Options

In addition to importing data from the above-mentioned instruments, FDM imports data from the following file formats:

- ASCII points file (user-defined formats allowed)
- Terramodel/GeodatWin (.pro)
- Roadline (.aln)

- Roadline 3D (.rln)
- Raw survey data (this requires an import .lgr file specific to the instrument the data was collected with.)
- TDS (.job)
- AutoCAD (.dwg, .dxf)
- LandXML (.xml)
- SiteVision (.ttm)
- MicroStation
- Landscape
- Roading 3D (.dc)
- USGS Digital elevation model (.dem)

Download/Import Data Format Template (a.k.a. .lgr or logger file)

When you import points or survey data, Terramodel will usually ask you to designate a logger file as well as import file. Sometimes, if you are downloading and importing in a single script, Terramodel assigns the logger file for you. Logger files, also called data format templates, are named for the type of data they translate.

Terramodel is shipped with a variety of data format templates. These templates are stored in the `Program Files\Trimble\Shared\Formats` folder. These templates may be updated or added between releases. If you have an instrument that does not have a template, contact the Terramodel Support Department or your local dealer. If you can provide an example of your instrument's output, the Support Department may be able to create a template for you.

Many data format templates have accompanying documentation that summarizes the applicability and limitations of the data format template. This accompanying documentation is also stored in the `Program Files\Trimble\Shared\Formats` folder.

Export File Format Options

In addition to uploading data to the survey devices listed in the following section, Terramodel exports data to the following file formats:

- ASCII points file (user-defined formats allowed)
- DC points (.dc)
- Terramodel/GeodatWin (.pro)
- Geodimeter Area (.are)
- Roadline (.aln)
- Roadline 3D (.rln)
- Roadline 3D (.dc)
- SDMS
- Softdesk Fieldbook (.fbk)
- STAR*NET
- TDS (.job)
- TDS (.dtm)
- AutoCAD (.dwg, .dxf)
- SiteVision (.ttm)
- MicroStation

Terramodel Upload Options

Terramodel supports uploading to most current Trimble devices and many older models.

TRIMBLE DEVICES

- TS315
- Trimble 3300
- Trimble 3600 (Elta, GDM, TDS)
- Trimble 5600 (Elta, GDM, TDS)

- Trimble Constructor 50/55
- Trimble DiNi (M5 format)
- Survey Controller
- Survey Data Card
- older devices including Geodimeter devices and Zeiss Elta (M5 format) devices

TDS DEVICES

- TDS Survey Pro CE
- older devices including TDS Husky and TDS running on an HP48

MISCELLANEOUS OTHER DEVICES AND INSTRUMENTS

- Leica GIF (a/n and gsi)
- Nikon DTM 310/500
- Pentax PCS 300
- Sokkia (SDR 20/33)
- SMI V6 or on HP 48 V5
- TPC on Husky

Complete export/upload capability is not offered for every device listed.

The devices with which Terramodel can communicate is updated regularly, and the list above is necessarily incomplete. If you do not find your instrument or data file format in these lists, please call your local distributor to see if it has been added recently.

The Import/Export Commands

The import command runs an import script. A script uses a series of dialog boxes to guide you through the download and/or import process.

There are two ways to execute the import command in Terramodel:

- From the File Menu
- From the Terramodel command line

Naming Scripts

As you create your own import scripts, you may name them as you see fit. Terramodel is shipped with several example scripts for commonly encountered data sources.

Import or Export Using the File Menu

Follow these steps to run an import script from Terramodel's File menu:

1. From the File menu, select *Download/Import* or *Export/Upload*. A sub-menu appears that consists of two sections: the *Import or Export Script Manager* option and a list of currently active scripts.
2. Select one of the scripts from the script list. Terramodel runs the script. The script's dialog boxes guide you through the download/import process. *Please see the Import-Export online help for specific information on these dialog boxes.*
3. The final dialog box of each import process is the *Import summary* (or *Export summary*) dialog box. This dialog box is not a script option; it appears at the end of each import process. This dialog box summarizes the instructions given to Terramodel for execution of the import process when running a script. Use it to verify data entry and settings before completing the import. Even if certain dialog boxes are not presented at runtime, the default settings, which the script provided, appear in this summary. You can abort the operation and edit the script, if necessary, before starting over.
4. Click **Import** or **Export** to perform the operation. When completed, the results appear at the end of the Summary report. Click **Finish**.

Note: Only the dialog boxes you elected to see by checking the Show this page at runtime check box display during script creation. Please see the Creating Import Scripts section on page 286 for more information.

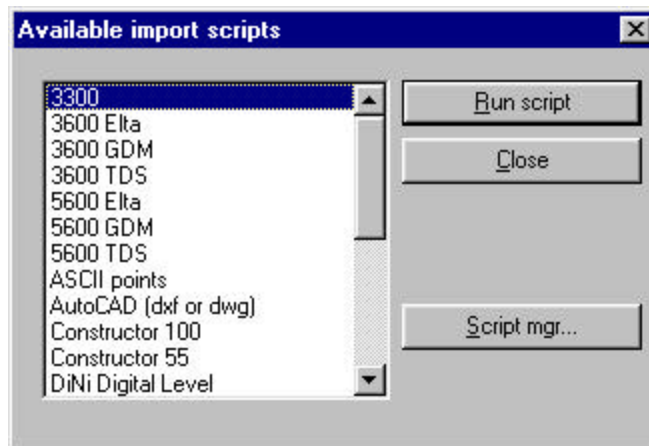
Script Navigation Buttons

Use the buttons at the bottom of each dialog box to navigate through the import or export script.

Button	Description
Back	Click Back to return to the previous dialog box in the script.
Next	Click Next to proceed to the next dialog box in the script.
Cancel	Click Cancel to abort the command and return to Terramodel.
Import/Export	Click Import (or Export) to perform the import based on current script settings.
Finish	Click Finish to complete the command and return to Terramodel.

Import or Export Using the Terramodel Command Line

You may also run a script by typing the **import** or **export** in the Terramodel command line. The *Available import scripts* (or *Available export scripts*) dialog box appears, displaying the active scripts. (To see a list of all the available scripts, use the script manager, where you can scroll through the whole list and activate or deactivate scripts.)



From the *Available import scripts* (or *Available export scripts*) dialog box you have the following options:

Button	Description
Run Script	Click the name of the script you wish to run; then click Run Script . Or double-click the name of the script to run that script.
Close	Click Close to close this dialog box and abort the import command.
Script mgr...	Click Script mgr... to enter the Import or Export Script Manager. Use the script manager to create, edit, activate/deactivate and delete scripts.

When you run the script, it will present you with one or more dialog boxes allowing you to enter the information necessary for the import or export process.

The final dialog box of each process is the *Import (or Export) summary* dialog box. This dialog box is not a script option, it appears at the end of each import or export process. This dialog box summarizes the script instructions. Use it to verify the settings before running the script import. Even if certain dialog boxes do not display at runtime, the default settings, which the script provided, appear in this summary.

Click the **Import** (or **Export**) button on the *summary* dialog box to perform the import or export process. When the process is complete, the results appear in the *summary* box at the end of the summary report. Click **Finish** to exit the Import or Export command.

Note: If you know the exact name of the script you wish to run, you may enter it after the import command. If the script name includes an embedded space character, the name must be enclosed in quotes. (ex: import "ASCII pts"). This will bypass the Available import scripts dialog box and immediately run a specific script.

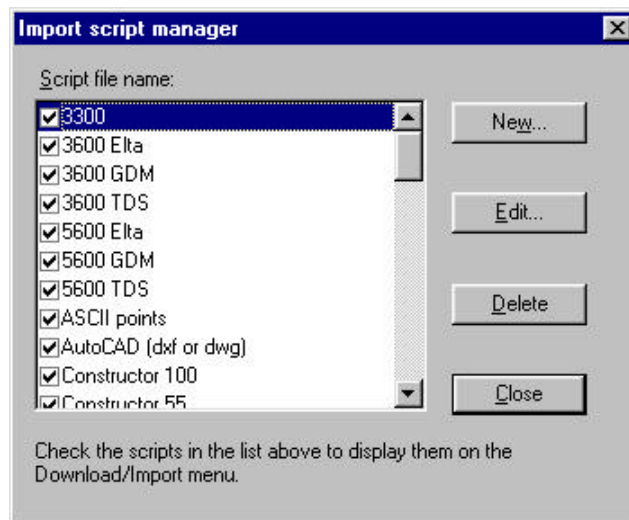
Script Navigation Buttons

Use the buttons at the bottom of each dialog box to navigate through the script.

Button	Description
Back	Click Back to return to the previous dialog box in the script.
Next	Click Next to proceed to the next dialog box in the script.
Cancel	Click Cancel to abort the command and return to Terramodel.
Import (or Export)	Click Import (or Export) to perform the import based on current script settings.
Finish	Click Finish to complete the command and return to Terramodel.

Script Manager

Use the import or export *script manager* to create, edit, activate/deactivate and delete import scripts. This is a screen capture of the *Import script manager* dialog box:

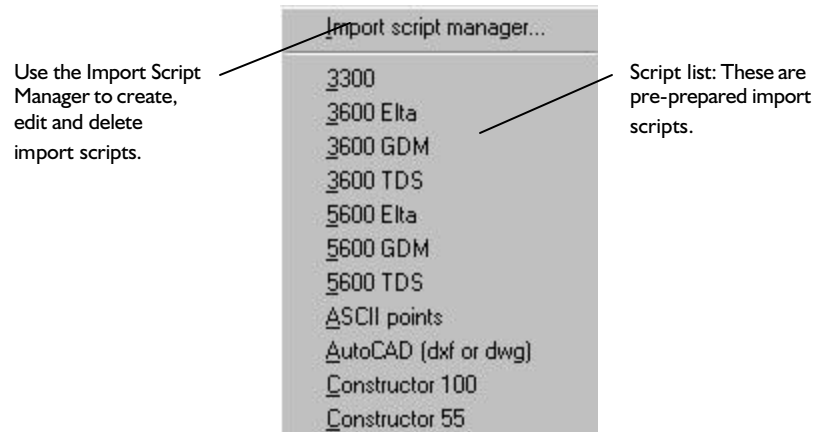


The *Export script manager* looks the same, displaying export scripts, and has the same features.

Creating Scripts

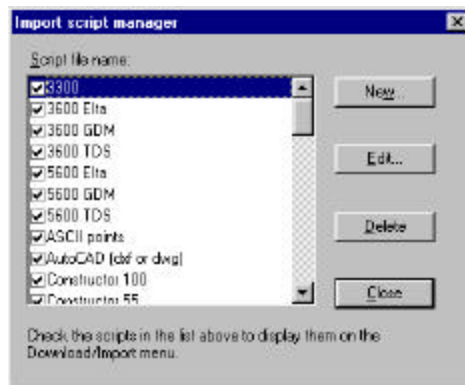
Follow these steps to create a script:

1. From the File Menu, select *Export/Upload* or *Download/Import*. A sub-menu similar to the following appears:



This sub-menu consists of two sections: the *script manager* option, and a script list. Any script created and activated by the script manager appears in the script list.

2. Select the Import script manager option. (You may type **ImportSMgr** in the command line in lieu of steps 1 & 2.) The *Import script manager* dialog box appears:



The names of all existing import script files, as located within the `Program Files\Trimble\Shared\ImportExport` folder, are listed in this dialog box. This includes scripts that are included with the original installation as well as any scripts you create. You may edit these scripts as necessary.

3. Click **New** to create a script.

For import scripts, the *Name script and select source/destination* dialog box offers the following options:

Action	Explanation
Download from instrument/data collector to disk	Terramodel copies a selected file from a connected survey instrument or data collector to a file on a destination disk. Terramodel asks for the filename to download, for communication parameters, and where to write the data file.
Download to disk and import	Terramodel downloads and saves the file to disk, and then imports the data in that file into the active Terramodel project. The dialog boxes used in this scripting process are a combination of those used in both the Download and Import processes.
Import from disk	Terramodel imports data from a selected file on disk into the active Terramodel project.

The export scripts, the *Name script and select source/destination* dialog box offers these options:

Action	Explanation
Export to disk	Terramodel exports selected data from the active Terramodel project to a file on disk.
Export to disk and Upload to instrument/data collector	Terramodel exports selected data from the Terramodel project to a file on disk and uploads the exported file to a survey instrument or data collector in one step. The dialog boxes used in this scripting process are a combination of those used in both the Export and Upload processes.
Upload from disk	Terramodel uploads a file on disk to a survey instrument or data collector. If you choose this option, a correctly formatted file must already exist on disk.

Type the name of the new script in the Script name control. Use a unique name that will let you know what it does and how to differentiate it from another script. The script name entered here subsequently appears on the script list.

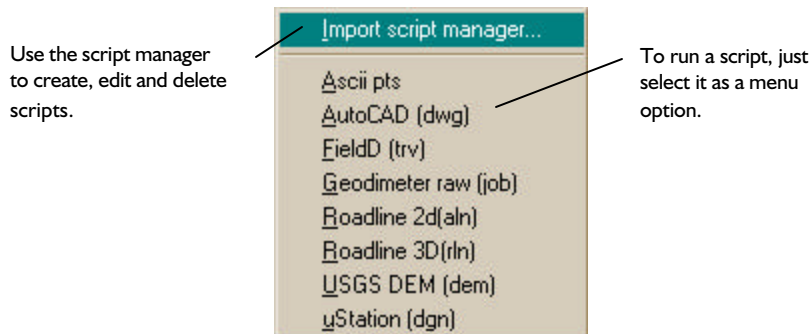
4. After you name the script and choose a the type of script, click **Next** to continue to the specific Download and/or Import script creation dialog boxes associated with that choice. *These dialog boxes are described in detail in the online help.*

Note: Please see the Creating Scripts vs. Running Scripts section for more information on how script creation or editing differs from running scripts.

Editing Scripts

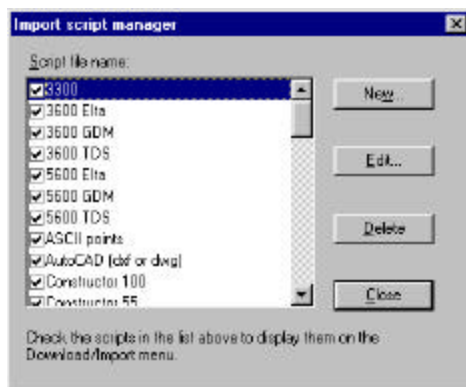
Follow these steps to edit a script:

1. From the File Menu, select *Import/Download* or *Export/Download*. A sub-menu similar to the following appears:



This menu consists of two sections: the *script manager* and the list of active import or export scripts.

2. Select the *script manager* option. (You may type **ImportSMgr** or **ExportSMgr** in the command line in lieu of steps 1 & 2). A *script manager* dialog box similar to the following appears:



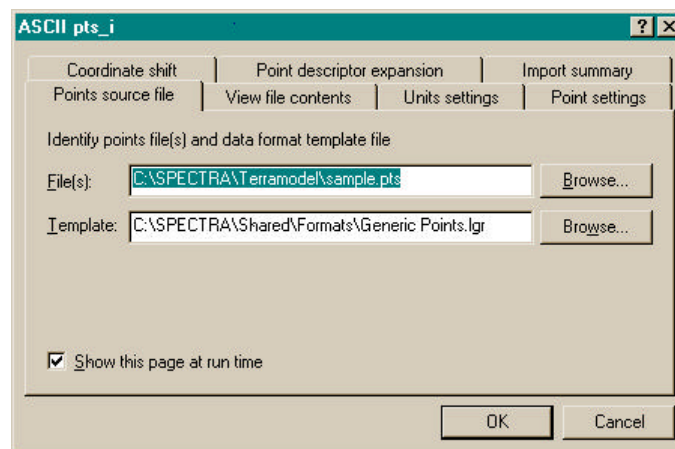
All scripts are listed in this dialog box. Any scripts with a checkmark beside them appeared as a menu option.

3. Select the script you wish to edit and click **Edit**.
4. A dialog box appears with tabs. These tabs display the titles of the dialog boxes displayed during script creation. Use the tabs to select the options you wish to edit. *Refer to the online help for more information on scripting options. Please see the Editing a Script - An Example below.*

Note: Please refer to the Creating Scripts vs. Running Scripts section on page 292 for more information on the differences between running scripts and creating or editing them..

Editing a Script – an Example

Each script presents dialog boxes to facilitate the download/import process. When you edit an import script, these dialog boxes appear as tabs within an edit dialog box. For example, the following dialog box appears if you edit Terramodel's sample ASCII *pts* import script:



The name of the script appears as the title. The tab titles vary according to the file type and source/destination options used in creating the script.

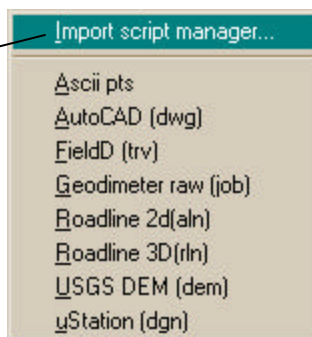
Use the tabs to view each script option. *Please refer to the Creating Scripts vs. Running Scripts section on page 292 for more information on the differences between running scripts and creating or editing them.*

Deleting Scripts

Follow these steps to delete a script:

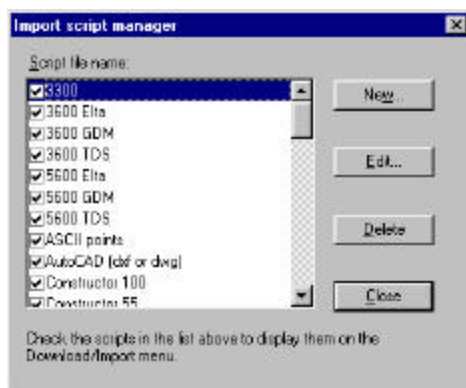
1. From the File Menu, select *Export/Upload* or *Download/Import*. A sub-menu similar to the following appears:

Use the script manager to create, edit and delete scripts.



This menu consists of two sections: the *script manager* option and list of currently active scripts, either import or export.

2. Select the *script manager* option. (You may type **ImportSMgr** or **ExportSMgr** in the command line in lieu of steps 1 & 2). The *script manager* dialog box appears, looking similar to this:



Existing scripts are listed in this dialog box.

3. Select the script you wish to delete and click **Delete**. The script disappears from the list.

Creating Scripts vs. Running Scripts

Before you create a script, think about the data you commonly import or export. Some elements of your import and export processes don't change. Other elements must be entered at runtime. As a script creator, be aware that many controls require data at runtime. If the script does not automatically provide this information, the script must present a dialog box to the user. When you create a script, there are three different input possibilities for each control in the script creation process:

- Leave the control blank, and check the *Show this page at runtime* checkbox so the user can enter the information at runtime.
- Enter a value in a field, and check the *Show this page at runtime* checkbox so the user can verify the default information at runtime.
- Enter a default value in each control, and uncheck the *Show this page at runtime* checkbox so that page of the script is not displayed at runtime. Those values can then be used automatically at runtime without displaying that page, and therefore without allowing user intervention. ()

The sections that describe each dialog box provide specific descriptions for all controls in the download/import or export/upload process. Some things to remember:

- Some controls are required at runtime. For instance, every import requires you to name an import file & data format template (*.lgr) file.
- Not all fields are required during script creation. Fill those which are likely to be standard for your routine import or export needs to streamline the import process.
- Dialog box appearance differs slightly at runtime from script-creation time. The script managers present a *Show this page at runtime* box during the script creation and editing processes, which is not shown at runtime.
- When you create a script, eliminate the dialog boxes you don't need by unchecking the *Show this page at runtime* box.

Chapter Nine

Raw Data Editor

Survey data is a collection of observations tied to one or more known locations. The art and science of surveying applies many tools to the two factors influencing a survey's reliability: precise measure of the known locations in a surveyed network and the calculation methods used to estimate the locations of all points in a surveyed network.

Terramodel approaches the tasks of storing and processing raw data differently from previous versions. In the past, importing raw data, processing it, and converting it into coordinates were three distinctly separate steps. **Terramodel** now enables you to directly download raw data from a survey instrument or data collector, or import an old traverse file, and store the information directly in a project file.

Terramodel's Raw Data Editor finds traverses, intersections and resections, computes coordinates, and performs coordinate adjustments for the imported raw data automatically. One of the coordinate adjustment options is a Network Least Squares adjustment. The Raw Data Editor also includes an interactive data editor that allows you to add new information or modify the current survey information.

The Raw Data Editor

The Raw Data Editor (RDE) is a **Terramodel** command. It collects all imported survey job data and displays it in a single format. The RDE includes an interactive editor that allows you to view, search, edit and delete raw data that is displayed in a consistent format independent of the data source. The RDE automatically converts the angles and distances of the raw observations taken in the field into best-fitting least squares coordinates. The RDE also allows you to control the settings and tolerances used during computation.

*Note: In this documentation, RDE is used to refer to a **Terramodel** command and as an abbreviation for Raw Data Editor. When it is used as a command, it appears in a font reserved for commands, like this (**RDE**).*

RDE Features

The RDE Uses Total Station Data, GPS (RTK) Data, and Level Data

The RDE normally uses raw data measured and recorded with a total station, but it is not limited to data collected in this way. By using the **Terramodel Import** command, survey information stored in a file in almost any ASCII format can be brought into the RDE. Raw data can be imported from a file regardless of its source. The RDE automatically finds and calculates traverses, resections and intersections, and (if the options are set accordingly) automatically adjusts the resulting coordinates by Least Squares variation of coordinates. The RDE does not need to be told how to compute any station. Backsight and foresight information and the order of station occupation are irrelevant. The RDE only requires that a *point ID* is recorded with each observation and that good surveying technique is used in the field.

The RDE allows you to import GPS Real-Time Kinematic data. This data consists of the vector difference between a local base station and the receiving instrument and the associated covariance matrix. If you import GPS data, you must define a local geodetic grid with the **GPSsys** command (*Settings/GPS Geodetic system* menu option) before you import and before you compute data with the RDE.

The RDE also processes level data, such as the data created by the Trimble DiNi digital level. This gives you the opportunity to gather very accurate elevation data. You can import level data or enter it by hand. The RDE automatically computes level data before any other gathered elevation data in order to preserve elevation accuracy.

Works with multiple data formats

The RDE performs computations on raw survey data imported into **Terramodel**. **Terramodel** imports data in many different ASCII formats. These include virtually all formats produced by modern total stations. It can also import survey information from data files created by other programs or files created manually. The RDE displays all these types of raw data in the same simple, easy-to-understand and user-definable format regardless of the source of the data.

If the format of your total station is not specifically listed in the Terramodel import wizard, it may be included as part of another format or it may need to be added. The import wizard has been constructed so that most formats can be supported by adding one file and without modifying the program. Call our technical support department to find out if your format is already available, or what is needed to add it.

Assigns Computation Levels to Data

The RDE allows use of three levels of computation for coordinating stations: Primary, Secondary and Tertiary. These levels are used for controlling which parts of a control network to compute first, second or third and can be used to control traverse routing.

An additional feature of levels is that stations on higher levels are treated as fixed locations when computing stations on lower levels. Alternatively, the whole network can be computed together using least squares and computation levels used just to weight the various parts of the network differently.

Initially each survey job is assigned a computation level during the import process. Subsequently each occupied station can be assigned its own specific level.

Each level – Primary, Secondary and Tertiary – is assigned separate computation settings and tolerances.

Allows Data Modification with Interactive Editor

With the RDE edit commands, you can change, delete or insert survey information. Since your original downloaded survey data is safely stored in a separate download file on your hard drive, you can modify the data displayed in the RDE without concern. You can update observations based on field notes or if you receive new information, or if you question the field data.

Provides Computational Feedback

The RDE computation process takes place *behind the scenes*. The RDE user can track the computation process through a series of messages. These messages include multi-angle reduction results, traverse and least squares reports and tolerance failures.

Field Data Reduction – Fast Start

Use the Download/Import command to import raw survey information. Exit the RDE to view the adjusted traverse. You're done. **Terramodel** displays all of the reduced data points, error ellipses and traverse legs.

The process described above seems too easy to be true, but it is entirely possible. The process of downloading, importing and reducing field data can be highly automated or you can step through each alternative.

Imagine if you had the perfect super surveyor sidekick to reduce your field data. You give him the field data and you get back the best possible solution, a complete report of what he found and a final drawing showing error ellipses points and line work. He makes any assumptions required to get the job done and tells you what they are. If you want to check his work, he keeps meticulous notes about every assumption and tells you about every problem along the way. You can step in and make any type of change and the entire job is updated automatically with no complaints.

The RDE has been designed to do all of this and more. Although it can be used automatically, you can control every aspect of the reduction. The following section provides a more detailed description of the process.

In order to be in control you need to understand most of the process. In order to understand the details you need to read most of this chapter. Don't forget, you are putting your name on the line.

It is vital that you confirm the process of reducing your data from your procedures to coordinates. After you know the ropes you can begin to take short cuts and operate the

process in autopilot. The following is a short roadmap of the entire process. The details of the process are described in this chapter.

Step 1 - Collect the data.

Most users can continue to collect data the same way they always have. Some users will be able to modify their collection techniques to improve their speed and accuracy.

Step 2 - Configure the RDE settings

The most typical and logical settings have been established as the defaults. You may want to change them.

In all honesty, you can't say if the settings are OK until you understand what they mean. There is no secret here. You just have to look at each one and make sure each one is set to how you want to do things. It is strongly recommended that your adjustment method be set to *Network least squares with computation level order*. It's a tough job but once you have done it, most of them will not need to be changed. (*Note: Make sure you change them in the prototype project.*)

Now you really have to think. What corrections do you want to apply? You need to know what corrections are being applied in the instrument used to collect the data and/or in the software. Just make sure you only correct for the same thing once!

Step 3 - Import the data.

The download import command is typically used to get the data into the RDE. The other method is to hand-enter the data. During the import process the data is translated into the format required by RDE. This requires that the appropriate script file be selected and tested with your typical data file. If the data is imported incorrectly you must either modify your collection method or the translation file must be modified by Trimble to support your method.

One benefit of the RDE is that data imported from multiple formats are translated to a single format for editing. This feature enables you to use different types of equipment and combine all of the data on the same job.

Note: The RDE may take from a few seconds to a few minutes to compute and display the data. The RDE does the entire job from reduction to calculating the final adjustments in order to give you a complete report.

Step 4 - Edit the data

Look for any <Invalid Data>, <Unknown Key>, or <Duplicate Key> tags and correct these as required.

Review the messages in the *Message pane* and correct these as required.

Step 5 - Create a report

Enable all of the reports and print a report of the project. Review this report and make any adjustments as required. You may wish to revise the job based on the report.

Step 6 - View the results

When you leave the RDE, Terramodel automatically zooms to the extent of all of the visible objects. All of the points and lines are “coded” by layer, color and name to present the reduced data in the most understandable format. You may wish to revise the job based on the computed results.

Advanced topics

After you have mastered the basics and absorbed the information in this chapter, you will want to tackle the more advanced topics.

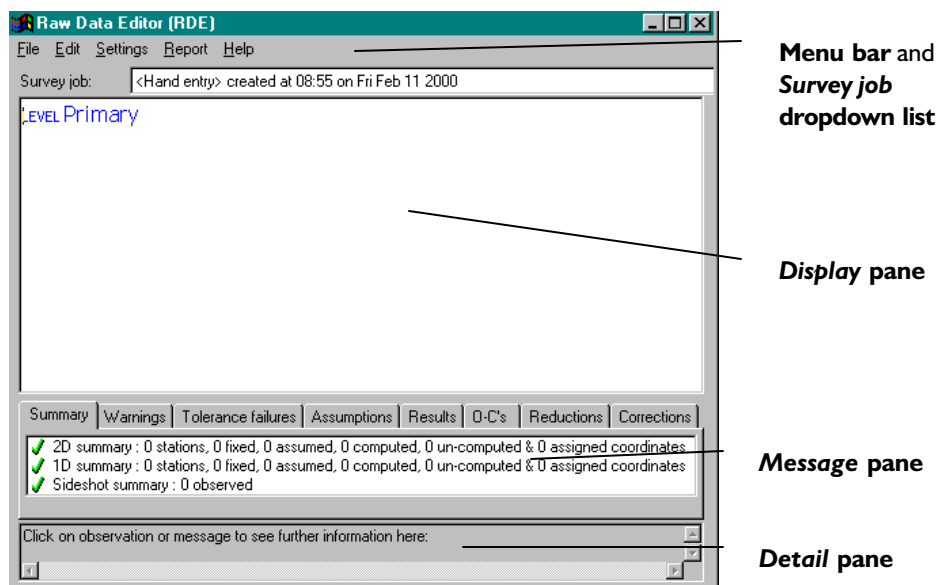
- Review your field procedures in light of the capabilities of the RDE. In particular if you have been doing only simple closed loop traverses you will want to begin to take redundant observations to take advantage of the least squares method. Review the section Survey Technique Suggestions on page 329.
- Create a written guide of how you will use the computation levels feature. This might be based on equipment or personnel.
- Create a written guide of how to use point names (descriptions).
- Create your own custom dictionary file so that RDE will use terminology that fits your standards.

RDE Dialog Box

Starting the RDE

The RDE allows you to add new information to raw survey data, to modify the current raw data, or to delete (or comment out) raw data you don't want to use. It also allows you to define the initial assumptions for the calculation process and to set tolerance definitions for three computation levels of raw data. As you enter the new information, the RDE recomputes (and adjusts!) the coordinates for all the stations and sideshots. When you exit the RDE, the raw data survey locations have been optimally calculated and already integrated with your project.

When you invoke the **RDE** command, the RDE dialog box appears. The body of the dialog box is divided into three distinct panes: the *Display* pane, the *Message* pane, and the *Detail* pane.



RDE Menu Bar

You can access all of the RDE functions and options from the RDE Menu bar and the Survey job dropdown list at the top of the RDE dialog box. The RDE File menu allows you to create or delete a survey job of raw data. Each survey job is edited separately although all survey jobs are calculated and adjusted collectively. The RDE Edit menu options allow you to modify the raw data and to create new data. The Tools menu options allow you to set overall project and display parameters and to set assumptions and tolerances for adjustment calculations. The Survey job control, a dropdown list, allows you to select a specific job of a survey project to display in the *Display* pane.

Menu Bar

Menu	Option	Function	
File	Create Survey Job	Creates a new job of survey data. Used to enter survey observations by hand.	
	Delete Survey Job	Deletes the current job of survey data. This is one way to completely delete points associated with raw data from a Terramodel project.	
	Exit	Exits the RDE with all coordinates calculated. <i>Note: When you return to the main program, the Undo buffer has been cleared.</i>	
Edit	<i>Note: All the options on the Edit menu work only with the information in the Display pane.</i>		
	Undo	[Ctrl]Z	Undo the last edit.
	Cut	[Ctrl]X	Delete the currently highlighted text and put it in a clipboard buffer.
	Copy	[Ctrl]C	Put the currently highlighted text into a clipboard buffer.

Menu	Option	Function
	Paste	[Ctrl]V If one line is on the clipboard, inserts the current contents of the clipboard buffer into the text at the current insertion point. If more than one line is on the clipboard, inserts the current contents of the clipboard buffer above the current line.
	Delete	[Del] Delete the currently highlighted text.
	Find	[Ctrl]F Display the <i>Find</i> dialog box. Then choose either the Point ID or Description data tag to search and enter a string of text to search for. If you want all survey jobs searched, check the box. The RDE finds the first instance of the indicated text and highlights the line.
	Find Next	F3 Finds the next instance of the Find string, moves the insertion point to the line and highlights the line.
	Insert Empty Observation	[Ctrl][Enter] Creates an empty line after currently active line. If using the keyboard equivalent, [Ctrl][Enter], the insertion point must be at the beginning or end of the line.
	Delete Observation	Deletes the observation on the line where the insertion point is.
Settings		
	Editor Settings...	Enter user-defined options concerning the project, the <i>Display</i> pane format, units, and precision. (See page 311 for more information.)
	Computation Settings...	Enter settings for general and level computation and tolerances. (See page 311 for more information.)
Reports		
	Reports	Select report data types and level of information. (See page 317 for more information.)

Survey Job Control

The RDE allows more than one set (or job) of survey observations to merge into one surveying project. A job of survey observations is often defined as the observations made by one surveying team over the course of a day's work. The Survey Job dropdown list allows you to select the survey job you want to display in the *Display* pane.

Display Pane

The *Display* pane occupies over half of the RDE dialog box, and displays the raw data of the currently selected survey job:

The screenshot shows a window titled 'Survey job: EXAMPLE.JOB imported at 11:50 on Thu Feb 24 2000'. The display pane contains the following text:

```

LEVEL Primary
JOB EXAMPLE PROJECT "EXAMPLE DATA" AIRTEMPERATURE 85.00F SURVEYOR BDH DATE 01-31-2000 INSTRUMENTMODEL 12345
OCCUPIEDSTATION 1 INSTRUMENT 5.38ft
REFSTATION 4 HA 00°00'00"
SIDESHOT 101 HA 354°07'32" VA 86°55'23" SLOPE 176.05ft TARGET 4.79ft DESCRIPTION CURB
SIDESHOT 102 HA 353°10'20" VA 86°38'43" SLOPE 152.14ft TARGET 5.00ft DESCRIPTION CURB
SIDESHOT 103 HA 351°45'29" VA 86°15'08" SLOPE 128.01ft DESCRIPTION CURB
SIDESHOT 104 HA 349°38'03" VA 85°50'03" SLOPE 103.03ft DESCRIPTION CURB
SIDESHOT 105 HA 346°24'27" VA 85°18'35" SLOPE 79.62ft DESCRIPTION CURB
SIDESHOT 106 HA 339°18'49" VA 87°28'14" SLOPE 53.30ft DESCRIPTION CURB
SIDESHOT 107 HA 278°38'51" VA 90°07'07" SLOPE 19.41ft DESCRIPTION CURB
SIDESHOT 108 HA 294°17'30" VA 89°11'48" SLOPE 8.18ft DESCRIPTION CURB
SIDESHOT 109 HA 306°04'49" VA 87°44'00" SLOPE 5.40ft DESCRIPTION CURBPOC
SIDESHOT 110 HA 298°56'00" VA 87°23'46" SLOPE 2.93ft DESCRIPTION CURB
SIDESHOT 111 HA 249°04'05" VA 90°25'42" SLOPE 3.88ft DESCRIPTION CURBPOC
SIDESHOT 112 HA 247°05'17" VA 91°40'27" SLOPE 7.89ft DESCRIPTION CURB
SIDESHOT 113 HA 258°45'37" VA 91°47'08" SLOPE 16.34ft DESCRIPTION -CURB
SIDESHOT 114 HA 195°32'31" VA 93°08'47" SLOPE 28.51ft DESCRIPTION +CURB
SIDESHOT 115 HA 169°18'15" VA 93°01'59" SLOPE 31.55ft DESCRIPTION CURBPC
SIDESHOT 116 HA 162°52'50" VA 93°13'13" SLOPE 37.56ft DESCRIPTION CURB
SIDESHOT 117 HA 171°09'22" VA 93°18'18" SLOPE 47.38ft DESCRIPTION CURB

```

Usually the first several lines of raw data are informational. In the example above, the first line sets the computation level for the file; as the RDE requires a level to be set for every survey job. The second line names the job, project, surveyor, date, and instrument model and states the air temperature.

The remaining lines of raw data are **observations**. Every observation occupies one line of the *Display* pane. Each observation has one or more pieces of data. A **data tag** identifies each piece of data. These data tags do not have spaces within them and are not case sensitive. When you hand enter the data for an observation, you can enter the data/data tag sets in any order. However, when the RDE interprets and formats the information, it always displays the data/data tag sets in a standardized order.

Data tags can be customized. Internally, the RDE uses a set of **key names** to identify each piece of data. These key names are associated with the data tags in a data tag dictionary file. The RDE uses a default data tag dictionary for the data tags you see in the example above. (See page 339 for more information about data tags, key names and data dictionaries.)

Every Occupied station can have multiple reference station observations and multiple sideshot observations. The reference and sideshot observations can be mixed.

To get a printed list of the raw data, use the Reports menu option. (See page 317 for instructions on creating a report.)

Display Pane Format

The *Display* pane has a limited user-defined format. You can select a proportional or non-proportional font. You can designate that the data tags appear in a contrasting color and in a smaller font. These user-defined selections become the RDE default between uses. (For instructions on how to select a simple display or an easy-to-read display, see the RDE online help.)

In addition, you can control the data tags displayed in the *Display* pane by creating a custom dictionary file. This allows you to look at raw data with the data tags you are most used to. (See the RDE online help for information about creating a custom dictionary file.)

Data Entry in the RDE

Entering data manually in the RDE is not like using WordPad or Word. Here are the rules for entering data into the RDE *Display* pane:

- The RDE doesn't interpret and reformat the line until you move off the line. You can move off a line by selecting another line, or by pressing **Enter** or an up or down arrow.
- Enter only the first three or four characters of the data tag – just enough to make the string unique. The RDE searches the data dictionary for the first occurrence of that string. *Remember: Data tags are not case sensitive.*
- Leave a space between the data tag and the data. And leave a space between data and the data tag that follows.
- If you enter fixed coordinates for a station, it appears at every occurrence of that station.

- For angular measurements with degree, minute, second units, enter it in a DD.MMSS format. The RDE automatically converts the data into DD°MM'SS" format. *No matter which format is specified, angles must be entered without spaces.*
- Don't enter units. The RDE automatically supplies them. The RDE also automatically displays data to a pre-selected precision.
- Start and end remarks containing spaces in the string with a double-quote mark.

Data Entry – an Example

Before you enter the lines below to practice entering data into the RDE, create a <Hand Entry> survey job with the *Create new survey job* option on the File menu. For now don't worry about the Level Primary line that appears. Just enter each of the lines as listed below. As you enter each line of text, it appears in the *Display* pane. When you press **Enter** the RDE interprets the data tags and data, and reformats the information.

```
REM "OPTIONAL MANUAL ENTRY OF BEGINNING COORDS
STA 1 N 5000 E 5000 Z 1000
OCC 1 INST 5.25
REF 4 TAR 5.00
REF 2 HA 287.5925 SLOPE 181.260 VA 88.4120 BEAR N20.30E
OCC 2 INST 5.13
REF 1
REF 3 HA 254.0335 SLOPE 211.235 VA 91.5405
OCC 3 INST 5.28
REF 2
REF 4 HA 265.1715 SL 165.28 VA 86.3205
OCC 4 INST 5.17
REF 3
REF 1 HA 272.3945 SL 274.735 VA 91.3940
```

As you enter this data into the RDE, line by line, you should see the following data in the Display pane (*Note: some lines are wrapped to fit on the page.*)

```

REMARK "OPTIONAL MANUAL ENTRY OF BEGINNING COORDS"
STATIONINFO 1 NORTHING 5000.0000 EASTING 5000.0000 ELEVATION 1000.0000
  OCCUPIEDSTATION 1 NORTHING 5000.0000 EASTING 5000.0000 ELEVATION 1000.0000 INSTRUMENT
5.25ft
  REFSTATION 4 TARGET 5.00ft
  REFSTATION 2 HA 287°59'25" BEARING N20°30'00"E VA 88°41'20" SLOPE 181.26ft
DESCRIPTION TP
OCCUPIEDSTATION 2 INSTRUMENT 5.13ft DESCRIPTION TP
  REFSTATION 1 NORTHING 5000.0000 EASTING 5000.0000 ELEVATION 1000.0000
  REFSTATION 3 HA 254°03'35" VA 91°54'05" SLOPE 211.24ft DESCRIPTION TP3
OCCUPIEDSTATION 3 INSTRUMENT 5.28ft DESCRIPTION TP3
  REFSTATION 2 DESCRIPTION TP
  REFSTATION 4 HA 265°17'15" VA 86°32'05" SLOPE 165.28ft
OCCUPIEDSTATION 4 INSTRUMENT 5.17ft
  REFSTATION 3 DESCRIPTION TP3
  REFSTATION 1 NORTHING 5000.0000 EASTING 5000.0000 ELEVATION 1000.0000 HA 272°39'45"
VA
  91°39'40" SLOPE 274.74ft

```

Errors Highlight

No matter what display you select or whether the data is hand entered or imported, the RDE highlights the line of any errors it finds in red and bolds the error, as you can see in this example.

```

OCCUPIEDSTATION 8 INSTRUMENT 5.35ft
REFSTATION 7 HA 0.0000 <Unknown Key> Sp 4400
REFSTATION 9 HA <Invalid Data> 361.5778 VA 89.8514

```

In the first instance **<Unknown Key>** indicates the RDE has not been able to find a data tag beginning with "Sp" in the current data dictionary.

In the second instance **<Invalid Data>** the RDE has found the data itself to be out of range or not recognizable.

To fix these errors, just edit the offending data tag or data. You don't have to remove the **<Unknown Key>** or **<Invalid Data>** tags. When you move off the line, the RDE re-interprets the line. If it finds no more errors, the RDE automatically removes the error message and the color returns to normal.

To Edit Data for an Existing Observation

1. Place the location marker on the observation you want to edit.
2. Enter the data tag and data you want to add. Make sure there is a space before and after the data tag and the data.
3. Move off the modified observation. The RDE interprets the modified observation, redisplay the observation line, and re-computes the coordinates.

To Enter a Remark Line

1. Place the location marker at the end of the observation line preceding the line you want to enter.
2. Press [Ctrl][Enter]
OR
Select *Insert Empty Observation* from the Edit menu.
3. Type REM for the data tag. If your remark has no spaces, then type in the remark. If your remark is more than one word, begin the first word with a double quotation mark and type your remark. You don't need to close the quote, the RDE will do that for you.
4. Move off the remark line. The RDE interprets the remark and redisplay the line.

Note: You can also enter a short remark anywhere within an observation line.

To Create a New Observation

1. Place the location marker at the end of the observation line preceding the line you want to enter.
2. Press [Ctrl][Enter]
OR
Select *Insert Empty Observation* from the Edit menu.
3. Enter a data tag for observation type: *Occupiedstation*, *Referencestation* or *Sideshot*, and the point ID.
4. Enter the associated data/data tags for this observation. For an Occupied station you may add an instrument height; for a Reference station, a horizontal circle reading, a vertical angle and a slope distance; for a Sideshot, a horizontal circle reading, a vertical angle, and a slope distance. You may want to add a description

and a target height for the Sideshot. Make sure there is a space before and after each data tag and the data. (See page 304 for an example of hand entry data.)

5. Move off the newly created observation. The RDE interprets the observation data, redisplay the line, and recomputes the coordinates.

To Ignore an Observation

When you ignore an observation, that single observation is not used during the computation of coordinates. If you ignore all instances of an observation, it becomes a point with no location. In **Terramodel** points with no location are not visible in plan view. The point associated with the raw data is not deleted from the project unless you actually delete the observation line(s) from the RDE *Display* pane.

1. Place the location marker at the beginning of the observation line you want to ignore.
2. Type Ignored before the station type data tag (Occupiedstation, Referencestation or Sideshot). Be sure there is NO space between Ignored and the data tag.
3. Move off the observation. The RDE interprets the data, redisplay the observation, and re-computes the coordinates without this observation.

To Create a Rough Reference Observation

When you change a Reference observation into a rough Reference observation, the RDE does not use the rough reference data during the computation of provisional coordinates or least square adjusted coordinates.

1. Place the location marker at the beginning of the reference station line you want to designate as a rough reference.
2. Type Rough before the reference station data tag (Referencestation). Be sure there is NO space between Rough and the data tag.
3. Move off the observation. The RDE interprets the data, redisplay the observation, and re-computes the coordinates without this observation.

To Delete an Observation

This is the only way to completely delete the point associated with a raw data observation entirely from the **Terramodel** project.

1. Select *Delete Observation* from the Edit menu.
2. The RDE deletes the observation line(s) from the *Display* pane and corresponding point(s) from the **Terramodel** project (unless the points existed before the raw data was imported/typed in), redisplayes survey data, and re-computes the coordinates.

Note: If a point existed before raw data was imported or hand entered, then it is not deleted with these steps. It reverts to a point object with a location and with no associated raw data.

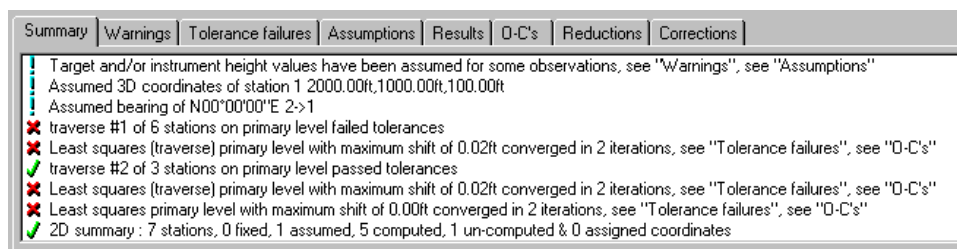
To Delete a Survey Job

This is the only way to completely delete a survey job from the **Terramodel** project.

1. Select *Delete Survey Job* from the File menu. The Delete survey job dialog box appears.
2. Select the survey job you want to delete.
3. If you want to delete the points associated with raw data in the survey job, check *Delete computed points*.
4. Click **OK** to proceed with the deletion. The RDE deletes the points associated with the raw data from the **Terramodel** project and redisplayes the RDE *Display* pane. Click **Cancel** to exit this dialog box without deleting anything.

Message Pane

The *Message* pane appears below the *Display* pane and displays messages generated during RDE coordinate computations. The *Message* pane (below) is generated when you first enter the RDE after importing the Example.job data from the Shared folder. (To do this, enter **Import Example (job)_i** on the command line. For more information about importing, see Chapter 3.)



During the coordinate computation, **events** (situations in the RDE) generate messages. Events can be information about the data, about the calculations made using the data or about the overall results of the RDE calculation process. The message itself comes in two forms: a brief message displayed in the *Message* pane and a more detailed message displayed in the *Detail* pane. The detailed message appears in the *Detail* pane when you select the message. The word *message* refers to the brief message.

There are several tabs in the *Message* pane and sometimes the same message appears on several tabs. For instance, a message that informs you that the RDE is assuming a particular coordinate location for the first station appears on the Summary Tab and the Assumption Tab.

To print a report of the Summary or Assumptions tab messages, use the Reports menu option. (See page 317 for more information on Reports.)

Message Pane Tabs

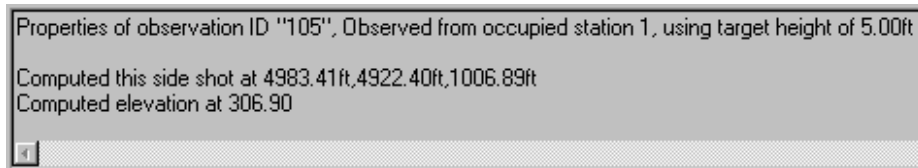
To help you navigate to the message information that you are interested in, the following table summarizes the types of messages that appear on each tab of the *Message* pane.

Tab Name	Type of Message Information
Summary Tab	Important messages. Often these messages are duplicated in one of the other tabs.
Warnings Tab	Any warnings from any step of the computational process.
Tolerance Failures Tab	Any tolerance failures during observation reduction, calculation of provisional coordinates, Least Squares adjustment, or calculating sideshots. (See page 331 for more information about the RDE computation process.)
Assumptions Tab	All assumptions the RDE has made during the observation reduction, calculation of provisional coordinates, Least square adjustment, or calculating sideshots. (See page 331 for more information about the RDE computation process.)
Results Tab	Traverse, Intersection, Resection, Partial Resection and Spur results. Least square results (converge/diverge). Information on stations whose location was not computed.
O-C's Tab	(Observed – Computed) values used during confidence tolerance tests in Least square adjustment.

Tab Name	Type of Message Information
Reductions Tab	Results from reducing redundant observations to one observation.
Corrections Tab	Corrections made to the raw data.

Detail Pane

The primary function of the *Detail* pane is to provide space for expanded message information. To see this expanded information, click on a message or an observation. The detail pane below appears when you click on the Occupied station “1” of the imported Example.job data. (To import this file, enter **Import Example (job)_i** on the command line. For more information about importing, see Chapter 3.)



When you select any station or sideshot in the *Display* pane, the coordinates and elevation (if computable) appear in the *Detail* pane. Many of the messages have detailed information. The messages on the Summary tab regarding a Traverse or Least Squares are particularly detailed.

To get a detailed report on Traverse or Least Squares results, use the Reports menu option.

Settings Menu Options

The RDE Settings menu allows you access to settings that affect the appearance of the RDE and control the RDE computation process. See the RDE online help for more details of the Editor Settings and Computation Settings tabs.

Editor Settings

Raw Data Editor Settings Tab

When you begin the **RDE** command, it searches the current Terramodel project for raw data objects. With the *Raw Data Editor Settings* tab, you can standardize the RDE's response to certain raw data conditions such as station or sideshot point location change, or two stations that are suspiciously close together.

Raw Data Appearance Tab

The RDE *Display* pane is designed to be very customizable. The *Raw Data Appearance* tab allows you to control the way RDE data and data tags appear in the *Display* pane.

Units Tab

The Units tab allows you to set the units the RDE displays. The settings you choose here only affect the way the data is displayed in the RDE. They will not affect the way the data is stored in Terramodel.

Computation Settings

The *Computations Settings and Tolerances* dialog box allows you to set the options, tolerances, and settings affecting the computational process of the RDE. The tabs on this dialog box allow the experienced surveyor to set the parameters for the RDE computational process. For the new user, the *General Settings* tab is the most commonly used. The other tabs allow the experienced surveyor to have detailed control over the computation and feedback process.

RDE Computation Level Feature

In addition to automatically processing the data in a traditional way, the RDE offers a feature called the computation level in its coordinate computations. There are three computation levels that can be set by the user: Primary, Secondary, and Tertiary. Each computation level has its own accuracy assumptions (set on the *Standard Errors* tab) and tolerance settings (set on the *Computation Level Tolerances* tab). Each survey job is commonly assigned to a Primary, Secondary, or Tertiary level.

The Sideshot computation level can be thought of as fourth computation level, although this level is only used internally. All sideshot observations are automatically assigned to this level.

For a project with multiple survey jobs, made by several survey teams, you can assign the smallest *Standard Error* values to the Primary level, and more lenient *Standard Error* values to the Secondary and Tertiary levels. When the RDE computes coordinates with a Least Squares adjustment based on computation level, the coordinates of the Primary level observations are computed without the Secondary level information. Then the Secondary level coordinates are computed using the Primary level coordinates as "fixed" locations.

In specific cases, the user can assign occupied stations to a specific computation level. The Sideshot level can not be assigned; all sideshots are automatically assigned to the Sideshot level.

You can also set the computation level for a single survey job. This allows the RDE to reference a more lenient set of accuracy assumptions and tolerance settings.

General Settings Tab

The *General Settings* tab is the most often used page of the *Computations Settings and Tolerances* dialog box.

Assumption Rules

If the RDE does not find a station with fixed coordinates within the survey data, it applies the following rules:

For raw data files for which the RDE can find a closed traverse: The assumed Northing, Easting and Elevation values are applied to the first occupied station in the traverse and the assumed Bearing value applied to the first foresight or backsight to an reference station. (See the section below on Computation Method/Route for more information.)

For raw data files for which the RDE can NOT find a closed traverse: The assumed Northing, Easting and Elevation values are applied to the first occupied station in the file and the assumed Bearing value applied to the first foresight or backsight to a

reference station. (See the section below on Computation Method/Route for more information.)

These user-defined assumptions can be changed at any time. Each time you update these assumptions, the RDE recomputes coordinates. When you add fixed coordinates to any station, the user-defined assumptions are no longer needed or used during the computation of coordinates.

Target Height Assumption - Single Target Method

During observation reductions, the RDE chooses a target height for each observation. Because target heights do not have to be recorded in the field it is possible that they can be forgotten and therefore missing in the data file. The RDE requires a target height to continue its computations. Therefore it assumes the target height(s) that it requires. There are two methodologies used for target heights. The *Multiple targets* checkbox controls which method the RDE uses.

When the *Multiple targets* checkbox is unchecked, the RDE treats all observations as if they were to a single target. In the single target method, the most recently observed target height in the data file is the one used for the observation regardless of whether the observation is to a reference station or sideshot. Here is an example of a the single target method:

	Observation	Observed Target Height	Assumed Target Height
OccStn STN1	HI 1.638m		
RefStn STN2	HA 327°12'21"	TH 1.750m	
RefStn STN3	HA 0°00'00"		TH 1.750m
RefStn STN3	HA 180°00'04"		TH 1.750m
RefStn STN2	HA 147°12'18"		TH 1.750m
SS 1	HA 206°05'43"		TH 1.750m
SS 4	HA 288°06'20"	TH 2.150m	
SS 5	HA 314°31'34"		TH 2.150m
OccStn STN2	HI 1.730m		
RefStn STN1	HA 19°12'55"		TH 2.150m
RefStn STN3	HA 326°27'20"		TH 2.150m
RefStn STN3	HA 146°27'37"		TH 2.150m
RefStn STN1	HA 199°12'53"		TH 2.150m

	Observation	Observed Target Height	Assumed Target Height
SS 10	HA 330°23'03"		TH 2.150m

Target Height Assumption - Multiple Target Method

When the *Multiple targets* checkbox is checked, the RDE treats observations made to reference stations as if tripods were established over them and observations to sideshots as if they were observed using a separate target.

This method is typically used while traversing with three tripods. Normally, a target height only needs to be noted for the observation to the foresight reference station, since the backsight reference station will previously have been assigned a target height. The RDE uses the last target height available to the reference station. The target heights for sideshots are treated independently from those for reference stations. The RDE assumes that observations to a sideshot are made to a separate target; changes made to the sideshot target height must be recorded on a sideshot.

The same raw data as above, when the multiple target parameter is checked, produces slightly different results:

	Observation	Observed Target Height	Assumed Target Height
OccStn STN1	HI 1.638m		
RefStn STN2	HA 327°12'21"	TH 1.750m	
RefStn STN3	HA 0°00'00"		TH 1.750m
RefStn STN3	HA 180°00'04"		TH 1.750m
RefStn STN2	HA 147°12'18"		TH 1.750m
SS 1	HA 206°05'43"		No target height
SS 4	HA 288°06'20"	TH 2.150m	
SS 5	HA 314°31'34"		TH 2.150m
OccStn STN2	HI 1.730m		
RefStn STN1	HA 19°12'55"		TH 1.750m
RefStn STN3	HA 326°27'20"		TH 1.750m
RefStn STN3	HA 146°27'37"		TH 1.750m

	Observation	Observed Target Height	Assumed Target Height
RefStn	STN1 HA 199°12'53"		<i>TH 1.750m</i>
SS	10 HA 330°23'03"		<i>TH 2.150m</i>

The following example of applying the multiple target rules to assuming target heights is a little more complex:

	Observation	Observed Target Height	Assumed Target Height
OccStn	STN1 HI 1.638m		
RefStn	STN2 HA 327°12'21"	TH 1.750m	
RefStn	STN3 HA 0°00'00"	TH 1.600m	
RefStn	STN3 HA 180°00'04"		<i>TH 1.600m</i>
RefStn	STN2 HA 147°12'18"		<i>TH 1.750m</i>
SS	1 HA 206°05'43"		<i>no target height</i>
SS	4 HA 288°06'20"	TH 2.150m	
SS	5 HA 314°31'34"		<i>TH 2.150m</i>
OccStn	STN2 HI 1.730m		
RefStn	STN1 HA 19°12'55"	TH 1.550m	
RefStn	STN3 HA 326°27'20"		<i>TH 1.600m</i>
RefStn	STN3 HA 146°27'37"		<i>TH 1.600m</i>
RefStn	STN1 HA 199°12'53"		<i>TH 1.550m</i>
SS	10 HA 330°23'03"		<i>TH 2.150m</i>

RDE - Observation Corrections

When observations are made they are made in the physical world on an irregular ellipsoidal shape (the shape of the earth); when they are calculated they are usually calculated on a uniform flat 3D grid. In order to convert the observations from the real world to the uniform grid, corrections are applied to the observations. These corrections are not calculated once, stored and used. They are calculated every time a particular observation is required, since the corrections themselves are dependent on the coordinates of the stations calculated using the corrected observations. In other words, the corrections and coordinates are calculated and used iteratively until neither the correction nor the coordinates are changing. *Not changing* for coordinates and linear observations is defined as less than one mm and for angular units as less than one tenth of a second.

Standard Errors Tab

The Standard Errors tab allows you to set a suite of expected standard errors for each computation level. In essence, these values are an indication of instrument accuracy, repeatability of the surveyor's technique and other related factors.

General Tolerances Tab

The *General Tolerances* tab allows you to set parameters affecting the overall Least Squares adjustment computations, a tolerance for target height variation during the reduction of observations, and several other miscellaneous parameters.

Rough Ref. Station Tolerances Tab

Traditionally, a surveyor uses observations to reference stations to orient and ultimately locate an occupied station. Once an occupied station is located, then the sideshots observed from that station can also be located. The RDE makes a distinction between observations made to reference stations that are to be used in the Least Squares adjustment, called Reference Stations, and observations made to reference stations that are used only to locate the occupied station with reference to sideshot observations, called Rough Reference Stations. Rough reference stations are not used in a Least Squares adjustment.

The *Rough Ref. Station Tolerances* tab allows you to set tolerances for the variance between a rough reference station location and the corresponding reference station location. Think of it like this: "I know I'm off a little, so I don't want to use this observation to this reference station in my calculations, but I want to know if my observations are off by too much." This tab allows you to set a value for "off by too much."

Computation Level Tolerances Tab

As it proceeds through the computation process, the RDE compares the tolerances that you set in the *Computation Level Tolerances* tab with the results of its calculations at specific phases of the computation process. If a calculated value exceeds the tolerance you set here, the RDE will notify you.

Note: the RDE continues to compute coordinates whether the tolerances you set are exceeded or not.

Report Options

The RDE offers several report formats to help the surveyor document their survey results.

Terramodel Graphics Interface

Each time the RDE computes coordinates, it also updates the **Terramodel** project. This section discusses the interface between the RDE and **Terramodel**.

Point objects

Terramodel creates a point object for each station (occupied or reference) and sideshot imported from a raw data file.

All associated data from the raw data file is stored in **Terramodel**. Unless location information is explicitly assigned to a raw data observation, i.e., unless the raw data observation is fixed, **Terramodel** assigns no location and no elevation to the point object as it is imported. The surveyor in the field can assign fixed coordinates to a station or the RDE allows the user to add fixed coordinates later. In addition, a station's location can be fixed during the import process itself.

Terramodel stores these point objects on a user-defined layer. Therefore, the user controls the selection, display and export with common **Terramodel** commands. For instance, if you want to change the color of all the raw data points, use the **Color** command.

Immediately after importing, these point objects have an undefined location. They are not visible in the **Terramodel** plan view window. After the RDE has calculated their coordinates, the points are visible.

Layer Assignment of Terramodel Project Objects

When raw survey data is imported into **Terramodel**, the default layer that it is assigned to is user-defined during the import process. When the RDE is invoked, it uses the following assigned **Terramodel** layers for the objects that it is computing. These layers are dedicated for use by the RDE.

Data Description	Assigned Layer	Object Type
Occupied station, Reference station, Sideshot	(User-defined during the import process.)	Raw Data Point object – a point object associated with raw data. Initially this point is without a fixed location.
Error ellipse	RDE_ELLIPSE	Block object (unit_cir.blk)
Traverse	RDE_TRAVERSE	Set object that connects stations defining traverse. Name: Traverse #x
Spur Traverse	RDE_SPUR	Set object that connects stations defining spur. Name: Spur #x
Intersection legs	RDE_INTERSECTION	Set object created by tracing legs from each occupied station to the station being intersected. Name: Point ID Intersection Legs
Resection legs	RDE_RESECTION	Set object created by tracing legs from occupied station to the station being located. Name: Point ID Resection Legs

Data Description	Assigned Layer	Object Type
Partial resection legs	RDE_PARTRESECTN	Set object created by tracing legs from occupied station to the station being located. Name: Point ID Resection Legs
Least squares network	RDE_LEASTSQUARES (initially established as an invisible layer.)	Total of all of the following: for each occupied station, a set object is created tracing legs to each reference object observed from that station whether or not the observation was used in provisional coordinate computation.

Importing Data and the RDE

Assign Fixed Location during Import

You can assign a fixed location to a station during the import process. During import, when **Terramodel** detects a raw data station with a matching point ID to a previously existing point object, it associates the existing point object with the station defined in the incoming raw data. In this case, the station is fixed at the coordinates previously assigned to that point object, if any are present.

To assign a Fixed Location During Import

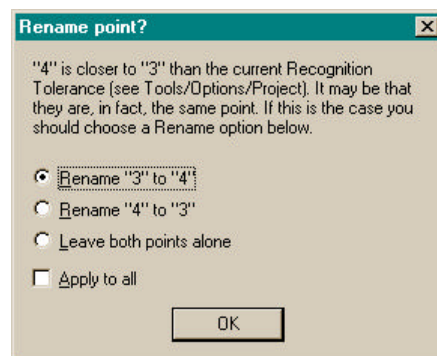
To assign a fixed location to a station during the import process, follow these steps:

1. Open a new **Terramodel** project.
2. Use the **Point** command to create a point object. Make sure the point ID matches the station ID in the raw data you want to import.
3. Run the **Import** command to import the raw data. You may need to create an import script with the **Importsmgr** command.

- Run the **RDE** command. The Raw Data Editor dialog box opens. The station with the ID that matches the point created in step 2 is fixed at the location of that point object.

Recognize Same Point

Some instruments or data collectors will not allow you to repeat a station ID. However, if you are closing a traverse, the first point and last point of the survey should be the same point. The RDE allows you to set a check for this situation. If you have checked the *Enable recognition* checkbox you may see the following dialog box appear when the RDE starts up:



Rename point Dialog Box Options

Select an option below.

Rename point dialog box directions. Select one of the following radio buttons.

Rename STNa to STNb	Reidentifies the first point ID to the second point ID to complete the traverse. Uses a single point in the network computations.
Rename STNb to STNa	Reidentifies the second point ID to the first point ID to complete the traverse. Uses a single point in the network computations.
Leave both points alone	Leaves both points as they are originally named and uses them both in the coordinate computations.

Apply to all

Check this box if you want the selection above to apply to all instances of close points the RDE encounters. If you do not check this box, the RDE will display this dialog box for every instance of coordinates closer than the *Recognition Tolerance* you defined. (See the RDE online help for more information.)

Terramodel Commands and the RDE

Terramodel is designed to prevent you from inadvertently deleting, moving or changing the elevation of point objects associated with raw survey data. Each station and sideshot point is linked to the raw data allowing **Terramodel** to provide a variety of defensive mechanisms. The first level of defense is to disallow the selection of a point associated with a station or sideshot within **Terramodel** commands that can move or edit it. You can also choose to disable this level of defense. Even if a point associated with a station or sideshot is selected and subsequently deleted or moved within **Terramodel**, the RDE has a second level of defense. If the RDE detects that a station or sideshot point has moved or been deleted, it asks for verification and further instructions. The possible variations of the verification process are discussed in detail in the RDE online help.

Surveying Basics With the RDE

Description of an Observation

An observation is made up of several pieces of data. Although there are many combinations of data, a typical observation consists of a horizontal circle reading, slope distance, vertical angle and Point ID. To the RDE every piece of data except the Point ID is optional (although normal survey procedures dictate specific data requirements). In the RDE a Point ID exists on every observation. Without knowing where an observation is from and to the other elements of the observation are useless.

Internally, the RDE uses key names to identify each piece of data. (See page 339 for the complete list of these key names.)

The following table lists the key names and comments for common observation data.

	Raw Observation Data
[Instrument Height] [Target Height]	Special care should be taken when measuring instrument and target heights. The RDE treats this data as if it is accurate.
[Horizontal Angle]	The RDE displays horizontal circle readings. The RDE uses horizontal circle readings of sequential observations to calculate the horizontal angle difference.
[Vertical Angle]	Vertical angles are normally measured as zenith (or elevation) angle.
[Slope Distance]	Distances are normally slope distances. However, some observations, such as observations made to a point of intersection, may not have a slope distance recorded.
[Horizontal Distance]	Horizontal distance is not easily measured. Since it is not clear what corrections (other than for slope) have been made to a horizontal distance, the RDE presumes that it needs no further corrections.

Sideshot observations can include data not used on reference station observations.

	<i>Sideshot Observation Data</i>
[Radial offset]	As it is unlikely that a surveyor could measure along the line of slope with any accuracy, radial offsets are treated as horizontal distances.
[Perpendicular offset]	Calculated at right angles to the direction of the observation. This measurement is also treated as horizontal distance.
[Elevation difference]	Elevation difference is not easily measured. Since it is not clear what corrections have been made to a elevation difference, the RDE presumes that it needs no further corrections. You can also use this to enter an elevation offset to a point directly above the observed station. Positive is upwards.
[Utility Elevation Difference]	You can use this to enter an elevation offset to a utility pipe directly below the observed station. Positive is downwards. The RDE presumes that it needs no further corrections.
[Remote Elevation Va]	The elevation of a point directly above a target can be calculated by observing a second vertical angle.

Types of Observations

A station is a point that is either occupied or is observed more than once.

Occupied station

An Occupied station is usually where the instrument is *set up*. This isn't a strict definition. For example, if a distance is imported into the RDE, the RDE needs to know the points the distance is between and one point must be an occupied station (even if the distance is measured with a steel tape).

Normally, an Occupied station has very few observation elements associated with it (other than instrument height). However the RDE accepts a computation level for an Occupied station. In this case, all observations from this Occupied station are used on that and lower computation levels.

Reference station

This type of observation is used for computing the coordinates of stations. These can be observed on face left and face right and using multiple rounds

Rough reference station

Rough reference observations are not used for coordinating stations; they are only used for calculating orientation prior to calculating sideshots.

Sideshot

Sideshots are used to observe points of detail. In the RDE, a sideshot can only be observed once. The RDE recognizes the second sideshot observation by the point ID. During the import process, a sideshot that is observed more than once either acquires a unique point ID or is upgraded to a reference station, depending on the RDE settings.

Target/Pogo/Rod/Range Pole

In order to calculate the elevation of the point on the ground it is necessary to record the height of the object the observation has been made to. Unfortunately there are several names for this in wide usage. You can create a custom data tag dictionary so that the RDE displays the word that is familiar to you. (*See page 339 for more information on data tag dictionaries.*)

The RDE has two separate schemes for target heights. One scheme assumes that all observations are made using one target. The second scheme treats reference stations as if tripods have been established. (*See page 313 for more information on assigning target heights using single and multiple target methods.*)

Station with Fixed Data

Often stations have pre-defined coordinates. These can be imported, entered by hand into the RDE or transferred from another project. When there is no fixed data the RDE will assume information. (*See page 312 for more information about applying user-defined assumptions.*)

Bearing

Bearing information is sometimes required to orient the stations in the survey within the physical world. Bearings can be imported or entered by keyboard into the RDE. When there is no orientation available the RDE will assume a bearing. (See page 312 for more information about applying user-defined assumptions.)

Description of a Traverse

There are two basic types of traverse – a closed traverse and an open traverse. Both have some similar requirements but there are also a number of variations on each type. Both traverse types must start and end on a previously positioned station, although for a loop or closed traverse this positioned station can be assumed. Each traverse leg must have been measured at least once. An angle must have been measured at each station that is not positioned.

A closed or loop traverse starts and ends on the same station. It can be orientated using one of the following techniques:

- Observing another positioned station not included on the traverse from the first station on the traverse.
- Assuming a bearing on the forward or backward leg from the first station on the traverse.
- Using a bearing supplied on any leg on the traverse.
- Using a bearing at any station on the traverse to another station.

An open traverse starts and ends on different positioned stations.

Both traverse types can be computed in either 2D or 3D. The positioned stations may be either occupied or unoccupied. If one or more of the positioned stations are unoccupied it means that no angle has been measured at the unoccupied station and therefore there is no angular balance.

Surveying Errors and Tolerances

Precision, Accuracy & Significant Figures

Precision is the measurement of how closely a series of measurements are to each other. If all the values are close to each other they are said to have high precision; if they are widely spread apart they are said to have low precision.

Accuracy is how close a certain measurement is to the true value. Accuracy is always unknown to some degree.

The number of significant figures displayed in the RDE is defined by the user and should not be confused with the precision or accuracy of the quantity being displayed.

Different parts of a surveyed network of points may be observed with more care than other parts. The RDE allows the user to place certain observations on different computation levels (*see page 295 for more information*).

It should be noted that a quantity may be measured very precisely and with considerable care but the resulting observation will not be accurate if the wrong quantity is measured. In one situation, a precise survey resulted in an unacceptable misclosure. The survey was repeated, again with a similar misclosure. The survey was repeated a third time with the same result. It turned out that one of the benchmarks used was in a wall that had been knocked down. The wall was rebuilt with the same bricks but the benchmark was not replaced in the same place. The three surveys were very precise, but not accurate because they were not close to the truth.

Care should be taken when selecting the number of significant figures to display the results to. They should reflect the accuracy of the observations.

Errors

There are three basic types of error: systematic, random and blunders.

Systematic errors are those which occur in a set pattern. Systematic errors can usually be accounted for in the process between making the observations and calculating the coordinates. For example, a surveyor uses a prism with a zero error and the measures of distances made with the prism are either too long or too short by the same fixed amount. Systematic errors are not necessarily observational errors. The failure to account for an appropriate geodetic correction can also be a systematic error. Systematic errors are dealt with by applying mathematical models.

A **zero error** is one that is always present and always the same. If a surveyor fails to notice that the first part of a steel tape is missing and records all distances as if measured from zero all the measurements made will have the same constant error.

A **scale error** is one that is dependent on the magnitude of the quantity being measured. If a surveyor uses a steel tape that has been stretched the distances measured will have a scale error.

Random errors are all those errors remaining in the observations after removing the blunders and systematic errors. It is possible (even probable) that some of these errors could be removed if a system could be established for them. However the fact that they remain means that they are considered to be random in nature and therefore they should be treated as such. In a Least Squares adjustment random errors are dealt with by applying probability theory. In a traditional adjustment the random errors are distributed according to some mathematical scheme.

A **blunder** is usually a mistake made by the surveyor such as observing the wrong target or misidentifying the correct one. A blunder such as a misidentified target will probably be obvious because of the magnitude of the resulting misclosures. Other blunders may be much harder to detect. If a target height is measured as 4.54 but recorded as 4.45, the resulting misclosure may be very hard to identify. Every effort should be made to eliminate blunders.

Least Squares

The RDE uses a mathematical approach to computing coordinates called *Variation of Coordinates*. *Variation of Coordinates* is a specific application of Least Squares adapted by surveyors to adjust coordinates to fit survey measurements. One of the most common applications of Least Squares is linear regression where a best fitting line is calculated through a series of coordinates. *Variation of Coordinates* is really only of use to the surveyor in conjunction with a computer. The process is not easily completed by hand because of the intensity of the mathematical calculations. A simple 10-station traverse results in hundreds of thousands of multiplications and additions.

The aim of a Least Squares adjustment is to adjust the coordinates of the stations so that the discrepancies between the final coordinates and observations, otherwise known as residuals, are as small as possible.

For various good reasons surveyors usually make more observations than is mathematically necessary to position the required stations. Additional observations are sometimes referred as *redundancies*. Traditional survey adjustments do not cope with redundancies very well but Least Squares provides an excellent method of incorporating these additional observations and applying probability theory to identify potential errors.

Unique Solution – the Benefit of Least Squares

One of the major benefits of Least Squares is that it can give a unique solution. If a series of interconnected traverses have been observed there can be many different routes through the stations. The problem with adjusting this sort of network using traditional adjustment methods is that there are as many different solutions as there are routes through the network. And as many different solutions for each route as there are types of traverse adjustment. There is no concept of a route in Least Squares. Therefore Least Squares produces one unique answer no matter how many interconnecting observations are made.

Survey Technique Suggestions

Because of flexibility of Least Squares and its ability to provide a unique answer, survey methods can change to take advantage of this adjustment method. The ability to identify non-systematic errors and the accuracy of Least Squares increases as the number of observations to each station increases. A traditional traverse has few redundancies, in part because of the time it takes to reduce the observations to coordinates by hand. Using the RDE allows you to gather redundant observations and increase the mathematical accuracy of your results.

The most important change to make in field technique is to observe as many stations as possible from each station. In addition, observe landmarks such as church towers from each station setup in the network. This immeasurably strengthens the reliability of the Least Squares computations.

Here are some suggestions for survey field techniques that will increase the RDE's computational accuracy:

1. Start observing on face left, unless there is a vertical angle to indicate otherwise. The RDE assumes face left on the first observation.
2. Include all stations to be observed at the setup in the first round of angles on face left.
3. When starting a new round of angles move the horizontal plate by more than the *HA shift* setting (and by less than 180° minus the *HA shift* setting).

4. Observe the stations in the following sequence:

Observed	Round	Face	Action
STN A	1	L	
etc...	1	L	
STN N	1	L	
STN N	1	R	
etc...	1	R	
STN A	1	R	
			<i>Move horizontal circle by 45°</i>
STN A	2	L	
etc...	2	L	
STN N	2	L	
STN N	2	R	
etc...	2	R	
STN A	2	R	
			<i>Move horizontal circle by 45°</i>
etc...			

RDE – Calculation Methods

Introduction

The RDE's primary function is to compute coordinates from raw survey observations. To do this, the RDE goes through the same stages once calculated by hand. First multiple observations from the same occupied station are reduced to a single measurement. Second, the RDE calculates provisional coordinates for each station using traditional surveying mathematics. This stage includes calculating a traverse route and adjusting it if requested, as well as performing intersections, resections and partial resections. The RDE allows the user to specify one of three traditional adjustment methods or no adjustment at all, to complete the provisional coordinate computation.

At this point the traditional surveyor has completed the task of calculating traverse coordinates. However, the RDE has an option to apply a Least Squares adjustment to the provisional coordinates calculated in the second stage. This creates a unique solution of coordinates for the entire network of stations. It can also highlight statistically odd or unusual results.

After the station locations have been adjusted (or not), the RDE computes the sideshot coordinates.

Each of these stages is described in more detail below.

Events and Messages

Events are generated during the computation process. An **event** is a situation in RDE that generates a message. This event can be information about the data, about the calculations made using the data or about the overall results of the RDE calculation process. The message itself comes in two forms: a brief message is displayed in the *Message* pane and, when the brief message is selected, more detailed information is displayed in the *Detail* pane. The word *message* refers to the message is displayed in the *Message* pane.

Step I: Reduction of Observations

For each Occupied station with observations to one or more Reference stations the observations are reduced to their mean values and the ranges of the observations tested against the user-defined tolerances. It is these reduced observations that are used everywhere else in the computations.

When any Reference station has been observed more than once from any Occupied station an event is generated. The detail information associated with this event contains a list of the raw observed and reduced observations. Two characters are added to the display of each raw observation that show the interpreted round and face.

All the events referring to observation reductions from a particular Occupied station are beneath the initial event for the reduced observations from the Occupied station.

An observation from an occupied station to itself is obviously not possible. In this case, the observation type is modified so that the observation is ignored and plays no further part in the computations. For this observation to be used, the ID of either the Occupied or Reference station must be changed and the ignored attribute removed.

Observations with Occupied Stations

Some instruments and data formats automatically record a Reference station and Horizontal circle reading along with the Occupied station information.

```
OccStn STN1   HI 1.510m
          RefStn STN2  HA  0°00'00"   automatically recorded
          RefStn STN2  HA  0°00'02" VA  90°07'22" SD 145.630m TH 1.510m
          SS 1         HA 106°25'00" VA  90°18'16" SD 12.190m Desc KC
```

These observations can cause some minor difficulties. In some cases these observations might be intended as a rough Reference station. If this is the case then the observation should not be used to coordinate stations. The template file that is used to interpret the raw data file may need modifying to ensure that these observations are not interpreted as observations to Reference stations.

If, however, these observations are to control Reference stations then there are two other problems. First the observation has no vertical angle. Second, the RDE will assume the observation is on face left, and generate an event to record the assumption. This is not

necessarily incorrect; however in many surveys the observation to the backsight needs to be defined more clearly. A horizontal circle reading alone is insufficient. Therefore a second observation follows the observation which is automatically recorded. The second observation is to the same Reference station and contains a vertical angle and slope distance in addition to the horizontal circle reading. This can make the automatically recorded observation redundant. If possible the automatically recorded observation is skipped over because it has been duplicated. This only happens for the first two observations following an Occupied station and only if:

- Both observations are to the same Reference station.
- The difference between the horizontal circle readings is less than the *Reduction Range /HA* user-defined value on the Computation Settings/Computations/Tolerances tab.
- There is a vertical angle on the second but not the first observation.

RDE Assumptions for Observation Reductions

Assuming face left: This event often occurs when an instrument or data format automatically records a Reference station and horizontal circle reading but no vertical angle along with the Occupied station information. An event is generated saying this has occurred unless the observation has been skipped. (See page 332.)

Assuming a horizontal circle reading: Normally the raw observations produced by an instrument do not contain horizontal angles. They usually contain horizontal circle readings that have to be subtracted from each other to give horizontal angles. However some data formats contain horizontal angles but not the horizontal circle readings that the RDE uses. The RDE interprets the horizontal angle as the horizontal circle reading to one of the Reference stations associated with the angle and a horizontal circle reading of zero is assumed to the other Reference station associated with the angle. The conditions for this assumption to be made are that the observation to the Reference station only contains the point ID, that there are only two observations to Reference stations from the Occupied station and that the two Reference stations observed are different. An event is generated for this.

Assuming target height: A target height can be assumed when the *Multi-target* method is selected (see page 312 for selection instructions) and no target height has been supplied for the first observation in a block to a particular reference station.

Range of observations

When reducing observations the RDE calculates the range of the observations. In some situations the *spread* of the observations is more commonly quoted but the range of the observations gives a clearer indication of the existence or otherwise of a problem with the observations.

Given the 4 values: 100.00, 100.00, 100.00, 99.00, the mean is 99.75. The spread is half of the difference between the maximum & minimum observed values which in this case is 0.5. The **range** of the observations is the maximum difference between the calculated mean and each of the observations. In this case this gives $99.75 - 99$ which is 0.75, 50% larger than the spread.

Standard deviation

The reason that the standard deviation of the observations is not used for testing the quality of the observation during reductions is that the size of the sample is usually small. The standard deviation becomes more useful as the sample size increases.

Tolerance Checks

User-defined range tolerances are checked during observation reductions. (*See page 317 for instructions on defining these tolerances.*) The RDE allows different range values to be designated on each of the three computation levels. The following table lists the observation elements and the tolerance against which they are checked.

<u>Reduction ranges</u>	Observation Reduction Tolerance Checks
	<i>The range of each observation element used in observations to reference stations is calculated during observation reductions. However the only the ranges for the observation elements that are used are checked against tolerances.</i>
[Horizontal angle]	The range of a horizontal angle is tested against the <i>Horiz. angle</i> tolerance specified for each computation level.
[Vertical angle]	The range of a vertical angle is tested against the <i>Vert. angle</i> tolerance specified for each computation level.
[Slope distance] [Horizontal distance]	The range of a slope or horizontal distance is tested against the <i>Distance</i> tolerance specified for each computation level.

[Elevation difference]	The range of an elevation difference is tested against the <i>Elev. difference</i> tolerance specified for each computation level.
[Bearing]	The range of a bearing is tested against the <i>Bearing</i> tolerance specified for each computation level.

Step 2: Calculate Provisional Coordinates

The RDE uses traditional survey methods to calculate provisional coordinates for each station. It automatically searches for a traverse route and any spurs (observations that connect or link to a traverse), intersections and resections. Using standard mathematics, the RDE calculates the provisional coordinates for as many stations as possible. If you have selected a Network Least Squares adjustment option, the RDE does no adjustment to the provisional coordinates. If you have selected a traditional adjustment, the RDE applies it to the provisional coordinates and moves to Step 4: Calculate Sideshots. (See page 312 for instructions on setting the adjustment option.)

Traverse

The RDE finds the routes for closed traverses, point-to-point traverses, and spurs without prompting from the user. In a complex set of survey points, the RDE may find a traverse other than the one you had in mind. You can control the stations the RDE considers for a traverse by assigning computational levels to specific stations.

When you select a Least Squares adjustment according to computation level, the RDE considers only Primary level stations for the first pass of coordinate computation. Then the Primary level station coordinates are assumed to be fixed for the Secondary level of coordinate computations. For the Tertiary level, the RDE assumes the Primary and Secondary level stations coordinates are fixed as it computes the Tertiary level station coordinates.

The RDE offers the following options to make an adjustment to any closed traverse. These are traditional survey adjustment methods.

No Adjustment

The RDE allows you to make no adjustments to the calculated raw data. This procedure calculates a provisional coordinate for all stations in a traverse, but applies no traditional adjustment to the provisional coordinates.

Compass/Bowditch

The RDE can calculate adjustments to a single closed traverse according to the Compass/Bowditch Method. This method adjusts the locations of all stations to close the traverse without a gap. In this coordinate adjustment method it is assumed the error along each leg of the traverse is proportional to the length of the leg.

Transit

Like the Compass/Bowditch method, the Transit method of traverse adjustment distributes the closing error among all the traverse coordinates. This method assumes the error is in the same proportion as the northing difference and easting difference of each traverse line is to the total northing difference and total easting difference of the whole traverse.

Crandall

The Crandall method of traverse adjustment assumes all angle measurements are correct and all error is in the length of the lines. This method uses that assumption and applies a simple Least Squares solution to calculate a solution with no change in angles.

Intersection

An intersection occurs when the target can not be placed directly on a station and the surveyor chooses to take observations of the point from two or more different locations. When a sideshot is intersected, the RDE can upgrade the sideshot to a station, calculate provisional coordinates, and include it with a Least Squares adjustment.

Resection/Partial Resection

The RDE calculates the location of a free station, i.e. an occupied station of unknown location, in one of the following ways. To perform a resection for the free station, the RDE needs the horizontal angles to a minimum of three known reference points. In this case the RDE calculates only the horizontal location of the free station. To perform a partial resection for the free station, the RDE needs the horizontal angle and distance to two known reference points.

Report stations not calculated

If the RDE can not calculate a station's coordinates, it sends a message to the *Display* pane.

Step 3: Calculate Final Coordinates with Least Squares

After the provisional coordinates for all stations, including intersections and free stations, have been calculated, you have the option of applying a Least Squares adjustment to the entire network. This general application of the Least Squares adjustment method is based on the principle that, if a set of measurements is made, each with the same reliability, then the most reliable solution is a unique set for which the sum of the square of the residuals is minimized. The Least Squares method makes a series of corrections called iterations. When the sum of the squares of the residuals decreases after each iteration, the solution is said to *converge*. This is good. If the sum of the squares of the residuals increases after each iteration, the solution is said to *diverge*. This means the Least Squares method is not finding a solution.

Results of Least Squares

There can be one of three possible results from the Least Squares adjustment:

- The solution set can converge, so the corrections are within the stated tolerance before the maximum number of iterations is completed. This means a unique solution set of coordinates has been found.
- The solution set can converge, but the corrections are not within the stated tolerance before the maximum number of iterations is completed. In this case you may want to increase the number of iterations or increase the correction tolerance. (This is very uncommon.)
- The solution set can diverge. When this happens, the Least Squares process will stop. There are two common reasons for this: the traverse misclosure is unexpectedly large or a station has been mis-identified.

First Iteration O-C warnings

On the first iteration the differences between the observations and the provisional coordinates are computed and tested against tolerances. This can be a useful if there is an error, which (perhaps due to poor network design or insufficient redundancy) is merely

distributed around the network and none of which final O-Cs (observed minus calculated) individually fail the tolerances or statistical tests.

For example, a traverse includes an additional check to a fixed station. When the traverse adjustment is carried out it is not possible to incorporate this additional check and so it goes unchecked. On the first iteration of the least squares the distance is checked and a message created. However due to poor network design the error is distributed among the other observations and no observation fails the statistical test.

Note: The first iteration O-C warnings are warnings only. They may only indicate the inability of the standard provisional computation methods to incorporate all the observations.

Final Iteration O-Cs, Residuals

After the last iteration the **residuals**, observed minus the final calculated value, are calculated. Their magnitudes are tested in two different ways.

Statistical test

Each residual is tested for statistical significance. It is divided by its own standard error and the result tested against the *Confidence* tolerance limit. (See page 317 for *setting tolerance instructions*.) The default *Confidence* value is 3. If an observation exceeded this value we could say with 99% confidence that the observation did not fit the network and was therefore in error.

Ideally the *sigma zero* value should be 1. If the *sigma zero* value is within tolerance the weights are modified by *sigma zero* so that *sigma zero* becomes 1. So when dividing the residual by its own standard error, the standard error may have been modified.

Tolerance test

Each residual is also tested against user-defined tolerances. This is an absolute test against fixed values. A residual that exceeds the tolerance is also marked with a cross in the *Tolerance* tab. You can effectively disable these events by setting large values in the *Tolerance* tab.

Step 4: Calculate Sideshots

After the station coordinates have been computed and adjusted, the RDE calculates the sideshot coordinates. The rough Reference stations orienting the sideshots are tested against a user-defined tolerance. (See page 316 for instructions on setting these tolerances.) If a station fails this test, the RDE sends a message to the *Message* pane.

Offset sideshot calculations

When field conditions don't allow a target to stand on the required point, the RDE can calculate the coordinates using offset data. For offset measurements taken along the same line as the survey instrument and target, RDE assumes a negative value for the offset distance indicates that the point lies between the survey instrument and the target. A positive value indicates that the point lies behind the target.

For offset measurements taken perpendicular to the survey instrument-target line, RDE assumes a point with a positive perpendicular offset lies on the right side of the target (looking from the survey instrument). A point with a negative offset lies to the left of the target.

The RDE can also handle two vertical angles. You can use these to measure the elevation of overhead cables. A sideshot target is set up directly below the cables you want to observe. The first vertical angle is to the sideshot target. The second vertical angle is to the cables above the target.

Key Names and Data Tags

Data tags identify each piece of data in the RDE *Display* pane. The user can define the data tags to conform to local surveying terminology. The RDE uses a dictionary file to allow the user to define the data tags to display.

Internally, the RDE uses key names to identify project and observation data. The key names available in the RDE are listed in the RDE online help along with a brief summary of function of each key name.

Chapter Ten

Reference Files

While editing a Terramodel project file, you can optionally display the data contained within one or more other project files. Those other files are referred to as *reference files*. You can divide your job into a number of project files, each addressing a different aspect of the overall design. As an example, while you edit the file in which sanitary sewers are designed, you can reference those in which the topographic survey, roadway and lot geometry, storm sewers, and water mains are contained.

Reference files provide a mechanism in which a team of people can collaborate on a project, each simultaneously editing a specific portion of your job's data, as contained within a particular project file, while optionally viewing other related project files, as desired. While others edit the files that you've referenced, after they save their work you'll be able to see those changes.

Terminology

Master Project

The *master project* is that project file that you open via the Terramodel **Open** or **New** command. When the master project is opened, any other project files that it previously referenced, before last being saved, are automatically opened as reference files. When it is closed, any project files that it references are automatically closed.

Reference Project

A *reference project* is a project file which is displayed in conjunction with the master project by reference. Data stored within the reference file is optionally visible, and can be searched, selected, and edited under certain conditions. However, that data will normally remain stored within the reference project, as opposed to being added to the master project. When you reference a project, only its name and certain associated *properties* are added to the master project.

Project Reference Prefix

When reference projects are loaded, there will commonly be objects of conflicting identity within the master project and the various reference projects. As an example, one or more of the master and reference projects may contain a point number 1 or record number 12485. The *project reference prefix* provides a convenient means of designating which project the object of interest resides in. It is intended as a means of concisely referring to a particular reference project or to the master project. A project reference prefix, sometimes referred to simply as a *prefix*, is a unique user assignable alphanumeric name, one of which must be associated with each project. It might typically be devised as short as possible, and is in fact limited to a length of four characters. The master project is given a default prefix of zero. The first referenced project is given a default prefix of 1. The second, 2, etc. you can reassign more meaningful prefixes if you desire. When you use the **ld** command to identify an object, the status bar at the bottom of the command bar identifies the prefix of the project within which the selected object resides. It does so by indicating **RefFile:** *prefix*, where *prefix* is the project reference prefix.

When during data entry you need to refer to an object by its record number or point number, it is presumed that you are referring to points or objects so identified within the active project. The use of prefixes to specify an object or point within a particular project other than the active project is not supported.

Active, and Background Projects

If we imagine a stack of drawings, drawn on a transparent medium, and lying on a drafting table, only the topmost drawing will be editable. In considering the set of files consisting of the master project and its reference projects in this same manner, only the topmost project can be edited. It is referred to as the *active* project. All other projects are

referred to as *background* projects. You will be able to designate which one of the projects is the active project. This process is referred to as *activating* a project.

Editable State

A reference project is said to be in an *editable* state when its entity data may be modified and saved to disk. If it is not editable, no entity modifications will be possible and the project will not be saved. A project is editable if it is not currently locked (editable) by another user, and if you have designated it as the active project.

Read-Only State

A reference project is said to be in a *read-only* state when that file is currently locked by another user. While in that state, you are prevented from activating the project and making it editable.

Read-Write Project

A reference project is said to be in a *read-write* state when it is not locked by another user. Master projects are always in a read-write state and editable, since a project that is locked by another user can not be opened as a master project. Read-write reference files can be made editable, which is to say that they can be activated, and their entity data can then be edited and saved.

Locked Project

Terramodel's file locking capability can be used to ensure that the entity data contained within an individual project may be edited by only one user at a time. To enforce this policy, those project files which are in a read-write state will automatically be locked while activated. While you have them locked, they will be in a read-only state from the viewpoint of other users on the network. When you activate another project, your previously active project will remain locked. You will be able to unlock it if desired, allowing other users to make it editable on their system. Your ability to keep a project locked, even if it is not your currently active project, allows you to reserve your read-write access to that project. This assures that you'll be able to activate it again without

risk that some other user will have locked that file. If, when activating a project, you no longer will need to keep the previously active project locked, you'll need to remember to unlock it, so other users can open it as a master project or activate it as a reference project.

Your master project will always be locked while you have the file open, and you will not be able to unlock it.

Properties of Reference Files

You can assign the following properties to a reference project, in addition to its project reference prefix as previously defined.

Visibility	You can make the objects contained within any non-active referenced file visible or invisible within the plan view. When you activate a project it is automatically made visible, and you can not turn its visibility off while it is the active project. See the section below entitled <i>Displayed Content</i> for an explanation of the control of object visibility within viewing modes other than the plan view.
Snap Setting	You can control Terramodel's ability to snap to objects contained within the referenced file. Note that Terramodel's running snap modes are not functional for snapping to objects residing within a reference project unless it is the active project. To snap to an object located in a nonactive reference project, you must specifically select an object snapping mode.
Color as Reference	Objects contained within the referenced file will normally be drawn using their individually assigned colors. You can, however, elect to display all objects within a particular reference project in a single color of your choosing. This feature can be used to help distinguish in which project file individual objects are located.

The Reference File Manager

Administration of reference files is performed by the *Reference File Manger* dialog box, as shown below. It is opened by the **RefFile** command. That command can be accessed by selecting **Reference files...** from the **File** menu.

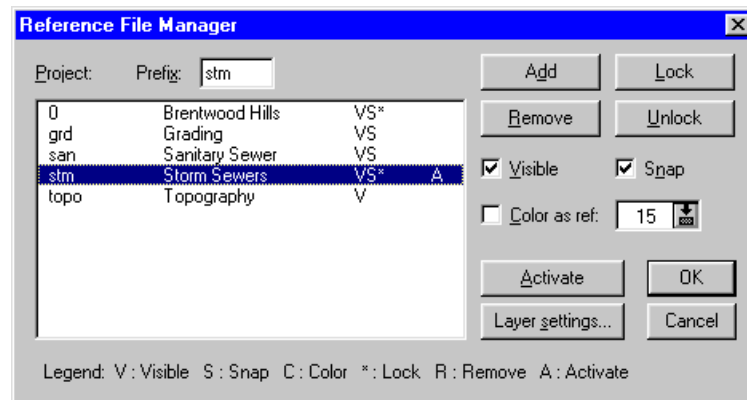


Figure 14–1 The Reference File Manager

This dialog box presents a list box showing the name and the project reference prefix of the master project and each of the reference projects. The list is sorted alphabetically by the prefix. It also displays the properties of each project, identifying that which is currently active, those which are locked, and noting those for which the visibility, snap, and color as reference setting is enabled.

You can add or remove reference projects. The *Add* button opens the *Select Reference File* dialog box from which you can select a project file to be referenced. After adding a reference file, you'll need to redraw the screen in order for its content to be displayed. The *Remove* button marks the currently selected project for removal, which is actually carried out when you press the *OK* button. It works in a toggling fashion to indicate the project for removal, or to remove that indication. After removing the reference to a project, you'll have to redraw the screen to make its objects disappear.

As a project is added, it is given a default integer prefix. If you wish, you can edit that prefix, which can contain up to 4 characters. You do so by selecting the desired project in the list and entering the desired text within the *Prefix* control. You may wish to supply meaningful prefixes, as shown in the above illustration, helping you to recall the appropriate prefix, based on the content of the project file, in referring to a particular project.

The *Activate* button allows you to establish the selected project as the active project. Double clicking on a project within the list box will activate that project, if possible, and close the *Reference File Manger* dialog box.

The *Lock* button allows you to attempt to lock the selected project. The *Unlock* button allows you to unlock a selected reference project (other than the active project) which you may have previously locked in order to reserve your access rights. The process of locking a non-active project file is discussed in more detail below.

You can control the selected object's *Visible*, *Snap* and *Color as reference* settings, as described above, within this dialog box.

The *Layer settings...* button opens the *Layer Settings* dialog box, from which the layer settings for the currently selected project can be edited. These layer settings in the master project are entirely separate from the settings in the referenced project. This enables you to display a project differently depending on whether it is a master project or a reference project. The layer settings are displayed, modified and saved without regard to one another.

Project Data Access Control

One of the benefits offered by Terramodel's reference file feature is the ability to share data in a collaborative manner, allowing multiple users to simultaneously work on the same job. This requires that no two users on the network be allowed to place the same project file in an editable state at the same time. That is prevented by locking the file. The rules involved in locking and accessing reference files over the network are described here.

File Locking

File locking can be enabled by checking the *Lock file on open* check box, within the *Server* group box of the *System Configuration* dialog box. The system configuration settings can be edited by selecting the **Config system/System** option (**system** command), from the **File** menu. You can find detailed information about **system** in the on-line help system.



It is STRONGLY suggested that every computer residing on a network maintain Terramodel's file locking in an enabled state. If you do not do so, the Terramodel project files that you open or reference will not be locked. In that case, other users on the network may be capable of editing the same file while you do so. This can result in loss of either person's edits as one user overwrites the work of the other.

When file locking is enabled, as you open a project, or lock a reference file, Terramodel creates a lock file of that same name, in the same directory as the project file, using a filename extension of `.plk`. The presence of such a lock file, related to the project that one user is trying to open or lock, indicates to Terramodel that another user has locked that file. This prevents that other user from opening that file as a master project, or from activating it in a read-write state. As the user who has locked a file closes that file normally, the lock file will be deleted. In the even of an abnormal termination, such as a power failure, the lock file may have to be manually deleted before any user other than that whom created it can again open its related project file. The user who created a lock

file, left as a result of abnormal program termination, can however, open the project file once again without having to delete the lock file.

The above described file lock setting affects only the creation of the lock file. Terramodel will not open a locked file, even if file locking is not currently enabled on the system.

The following discussion assumes that file locking is enabled on all systems on your network.

Opening a Master Project

When you attempt to open a master project, if it is not currently locked by another user, it will be opened in a read-write state and will automatically be locked by you. While you have that master project opened, it will remain locked, and no other user will be able to open that file. Other users will be able to reference it in a read-only state, but they will not be able to activate that reference file while you have it locked. You will not be able to open a master project which is currently locked by another user.

Activating a Project

When you attempt to activate a reference project, Terramodel will determine if it is currently locked by another user. If so locked, you will be prevented from activating that project. If it is not locked, it will be activated, and will be editable. No such determination is made when you reactivate the master project, since if you have successfully opened it, it will remain locked by you.

As a reference project is activated, it will automatically be locked by you so that other users will not be allowed to open or activate it. As this occurs, the reference file will be reloaded, so you be able to see changes made by other users since you last loaded it.

When you activate a project, the previously active project will remain locked. If you wish to make it available for editing by others on the network, you must unlock it. As described below, you can reserve your access rights to that or any other non-active reference project by locking it.

Locking Non-active Reference Projects

If while editing an active reference project, you find that you need to temporarily activate another project in an attempt to make it editable, you may wish to also retain your lock on the currently active project file. This will prevent another user from locking that project and preventing you from reactivating it, as required for you to continue

editing it. When you activate the other project, Terramodel will automatically retain your lock on the previously active project. If you wish to release your lock on a project after deactivating it, you may do so, by selecting and unlocking it. You should take care not to inadvertently retain a lock on a project that you do not need to reserve for your use.

You can also lock a reference project other than the active project, as long as it is not currently locked by another user.

Displayed Content

Within the plan viewing mode, the visibility of the objects contained within each individual project is controlled by the current visibility setting for that project, as established in the *Reference File Manager* dialog box. Within the other viewing modes, however, only those objects as contained within the currently active project are displayed. They are always displayed, since activating a project, automatically sets its visibility setting on.

Updating Reference File Content

While you are working, another user on the network is free to edit projects that you have referenced but not locked. If you wish to view the latest versions of those projects you have two options. If you can successfully activate and therefore lock a reference file, that file will be reloaded. Otherwise, you must reopen your master project. In doing so, all reference files will be reloaded as well. Note that in reopening the master file, all locks that you may have previously secured on reference files will be dropped.

Copying Data Between Projects

You can elect to copy selected objects from a source project into the active project using the **pcopy** command. The source project can be a reference project, or the master project, if that is not the active project. That command can be accessed by selecting **Project copy** from the **Edit** menu. It presents the following command bar.

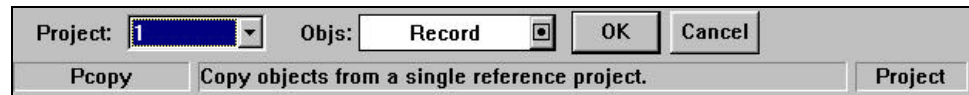


Figure 14–2 The Pcopy Command Bar

The *Project* combobox, presents a list of the source projects from which you can choose the one from which you wish to copy objects. That list is comprised of all projects, master and reference, other than the active project. You can then select from within the *Objs* select control, the objects that you wish to copy. You can only select objects that are located within the designated source project. You can do so, regardless of the current snap setting related to that project. You can even select a source project in the *Project* combobox which is not currently visible. After doing so, you can perform a redraw to make the objects in that project temporarily visible, so they can be selected graphically. Doing so will not affect that project's visibility setting, and those objects will again be invisible, after this command is completed, once you redraw the screen again.

When you press the OK button, the selected objects are copied into the active project. As a copy operation, the original objects remain within the source file as well. When an object being copied is on a layer that does not exist within the active project, that layer is created there. Objects therefore always remain on a layer of the same name in which they reside in the source file.

When an object being copied is represented by a linetype definition that does not exist within the active project, that new linetype is thereby defined within that project. When a linetype definition of the same name is found within the active project, the object is represented using that linetype definition, as defined in the active project, even if it differs from that linetype definition of that same name as defined within the selected project. Objects therefore always continue to be represented using the same linetype name with which they are represented in the source project, though the actual line pattern can vary if those linetype definitions are not consistent among the source and active projects.

As points are copied into the active project, integer point number conflicts are resolved by renumbering the points. If in attempting to copy an alphanumerically numbered point into the active project, an existing point with an identical alphanumeric point number representation is encountered there, that point will not be copied. *No warning is provided to indicate the failure to copy such points.*

As an object is copied to the active project, if it is referenced to another object, and if that object is also copied, the intended reference to the other copied object is maintained. If the object that it refers to is not also copied, the reference record will be reset to zero.

As an object is copied into the active project, its group number will be retained. If the active project contains objects within an existing group of that same group number,

Terramodel will associate the copied object with that group, as opposed to forming a new group with a different group number.

As an internal block object is copied into the active project if no internal block of that name currently exists there, such an internal block definition will be created within the active project. An instance of that block definition will be created at the location occupied by the block that was copied from the source project. If when copying an internal block, a block definition of that name already exists within the active project, the copy will be an instance of that block as previously defined within the active project. It is therefore possible that the copy block object may not be identical to the source block object.

As an externally defined block is copied in the active project a new instance of that block is simply created.

As an object is copied into the active project, any attributes assigned to that object are maintained, and those attribute definitions are created within the active project.

When a text object is selected to be copied into the active project, if it includes an EAT code containing a reference to a subject object, that subject will also automatically be copied into the active project, even if it was not explicitly selected. This is required in order for the EAT code to properly evaluate. If the EAT code contains a reference to a HAL, as required to label the station and offset of an object with respect to that HAL, the HAL object will also be copied to the active project.

The **pcopy** command, in combination with reference files, provides a convenient alternative to the use of the **tmxout** and **tmxin** commands, which will be abandoned in future versions.

General Conditions

The following general conditions and limitations apply to the use of reference files.

- Only those objects within the active project are editable.
- With the exception of layer settings, only those settings within the master project are editable.
- A project file must stand alone. No operation may be performed that would result in a project file which can not be opened by itself, without certain reference files. As an example, you may not create a set in the active project, between points that reside in other projects.
- A master project can not reference itself.

- Nested references are not shown. If a reference project references another project, the objects in that other project are not displayed.
- Raster images loaded within a reference project are not shown.
- TML commands can only react with objects located within the active project.
- A layer list defined in a reference project, can not be assigned to a dynaview created in the master project.
- Layer list definitions may not include member layers contained in a referenced project.
- The RefFile command can not be run when any other command is in use.
- Running snaps are nonfunctional when snapping to objects in a reference file. A specific object snap must be selected in all cases.

Chapter Eleven

Designing with Road, Pipe, & Hydro

How to Use the Roadway Commands

This section describes the general concepts of how you can use the **Roadway Design** module, hereinafter referred to simply as **ROAD**, to design a roadway. Most of the commands for this module are in the **ROAD** menu and are secured to this module.

Understanding Roadway Concepts and Terminology

A **Terramodel** project file might contain the design information for a single road, or for any number of roads. Road designs are done using some information that is specific to the design of a particular road, and some information that is general in nature and applicable to the entire project file, and perhaps even stored in libraries so that it can be applied to any project file. This is done so that standard design features can be quickly and easily incorporated into any road design.

Terramodel stores the following types of general design information which is used throughout all of the road designs in a project file:

- Materials.
- Surfaces.
- Template shapes.
- Design Settings.

Terramodel also stores the following design information which is specific to a particular road:

- Horizontal Alignments.
- Vertical Alignments.
- Road job.
- Roadway.
- Templates.
- Subgrades
- Cross-sections.

Each of these different types of design information are defined in the following pages.

Material

A *material* is a ROAD object that is used in many of the design decisions and calculations to identify the composition of a particular subgrade soil strata, or a roadway construction material. By defining different materials, **Terramodel** can accurately model cross-sections and individually accumulate the volumes of each material category. You will typically create a material definition for each of the subsurface soil materials encountered in your project, and for each of the pavement, based and subgrade materials used in the construction of your planned roadway. Note that as a minimum a Fill material is required and therefore a material of this name is automatically created the first time you enter the material manager. You can rename this default material as you wish.

The attributes assigned to materials include:

- ⊕ Shrink/swell percentage.
- ⊕ A color to let you visually differentiate between the various types of materials in the project, as you view the cross-sections.
- ⊕ An indicator as to whether or not the material is suitable for placement into structure fill.

You can find more information about roadway materials in **ROAD Menu**, “Materials”, in the Terramodel online help.

Surface

A *surface* is a ROAD object that is used to define the existing ground surface, the top of any defined subsurface material strata, and the finished design surface. Internally, the surface is modeled in the form of cross-section data at specific stations along an alignment. This allows surfaces to be defined either directly from measured cross-sections, derived by cutting cross-sections through a DTM, or a combination of both methods. A surface definition is implemented simply as a **Terramodel** layer, to which additional attributes and data are associated as described below. The volumetric extent of a subsurface soil strata is measured from its defined surface (the top of the strata) to the adjacent underlying surface, if any. A ROAD surface can therefore be thought of as defining both a surface or a volumetric region.

A surface can be defined as an *elevation surface*, where the elevation values represent the absolute elevation, or as a *depth surface*, where the elevation values represent the relative depth beneath the top surface. A surface such as a groundwater table surface can be designated as a *reference surface*, thereby being displayed in cross-sections for reference, but not affecting the design calculations.

One surface that must always be defined is the Finish surface that represents the finished roadway design surface.

The attributes that are assigned to a surface include:

- The material of which it is comprised.
- The type of surface, i.e., elevation, depth, reference or one sliced through the DTM contained on that surface's layer.

You can find more information about roadway surfaces in **ROAD Menu**, “Road Design/Road Job Surfaces”, in the Terramodel online help.

Shape

A *shape* in **Terramodel** is a building block for creating a roadway template. Each shape is composed of two or more control points, basic CAD properties, and some limited design information including general rules on how and when to use the shape. You might create shapes to represent a pavement surface, a concrete curb, a subgrade, a ditch, an embankment slope, or numerous other features. To create a shape, you use **Terramodel's** special graphical **Shape Editor**.

You can find more information on shape properties in **ROAD Menu**, “Shapes/Shape Editor”, in the Terramodel online help.

Shape Class

Shape classes let you organize your shapes into categories (like ditches, pavements, shoulders or medians) using a simple library. Intelligent use of shape classes will make reports, plots and generated DTM surfaces much more accurate and clear.

You can find more information on shape classes in **ROAD Menu**, “Shapes/Shape Class Manager”, in the Terramodel online help.

HAL

A ROAD *HAL* is a horizontal alignment consisting of a set or polyline object in the plan view that is registered in the ROAD HAL Manager. In registering the alignment as a ROAD HAL it is given a name, and optionally a set of alignment offsets and station equation data. The ROAD HAL can therefore represent an alignment that is offset from the actual horizontal alignment object with which it is associated.

You can find more information about registering HALs in **ROAD Menu**, “Alignments/HAL Manager”, in the Terramodel online help.

VAL

A ROAD VAL is a vertical alignment polyline object in the profile view that is registered in the ROAD VAL Manager, where it is given a name, and optionally a set of vertical offsets, allowing the ROAD Val to be offset vertically from the actual vertical alignment object with which it is associated.

You can find more information about registering VALs in **ROAD Menu**, “Alignments/VAL Manager”, in the Terramodel online help.

Design Settings

The ROAD design settings are used by **Terramodel** to determine how to compute and display the results obtained from the design input. Among other things, these settings let you toggle the ability to:

- Dynamically show the interaction between the original ground and a template.
- Apply shading to template shapes.
- Use the template data in the design.
- Use subgrade data in the design.
- Use transition data in the design.
- Use superelevation data in the design.
- Use phase data in the design.
- Apply curve correction to the design.
- Display user-defined grids in the template and cross-section editors.
- Horizontally and vertically position templates based on the roadway HAL/VAL or finish cross-sections.
- Vertically position subgrades based on the roadway VAL or finish cross-sections.

You can find more information about road design settings in **ROAD Menu**, “Settings/Design Settings”, in the Terramodel online help.

Road Job

A *road job* is a named definition of the collection of data that pertains to a particular road’s design. A road job can consist of one or more individual *roadways*. As an example, the southbound and northbound portions of a road job can be modeled as distinct roadway, each having their own horizontal and vertical alignments if need be. A road job is always assigned a main horizontal alignment, from which the roadway stationing is

derived. It may or may not be that associated one of the designated roadways. This alignment is a registered ROAD HAL, as described above.

You can find more information about road jobs in **ROAD Menu**, “Road Design/Road Job Manager”, in the Terramodel online help.

Roadway

A *roadway* is a named collection of data that pertains to a particular set of related traveled ways. A roadway can define all of the lanes traveling in both directions, or individual roadways can be defined for the lanes traveling in either direction, or ramps, access roads, frontage roads, etc. A roadway is associated with both a horizontal and a vertical alignment. These must each be registered HALs and VALs as described above.

A single roadway (such as the northbound lanes) in a road job consists of a name, zero or one (1) horizontal alignments, and zero or one (1) vertical alignments. When you first create a road job, **Terramodel** also creates a roadway with the same name.

You can find more information about defining roadways in **ROAD Menu**, “Road Design/Roadways/Roadway Manager”, in the Terramodel online help.

Finish Template

A *finish template* in **Terramodel** consists of one or more shapes that are combined and attached to each other to describe the cross-sectional configuration of the finished roadway surface at the particular station at which that particular configuration starts. In constructing the template **Terramodel** can combine particular shapes together, depending on whether the shape is in cut or in fill, or depending on the subsurface material strata within which it lies. You can designate these limitations as properties of the shapes as they are defined. You can designate alternate shapes to be used in those cases where a shape’s required conditions can not be met. **Terramodel** supports numerous powerful mechanisms for transitioning from one template to another.

You use the graphical **Template Editor** to build a template from shapes. You designate which shapes respond to superelevation, and then define and apply the transitions for the template. You can find more information about building finish templates in **ROAD Menu**, “Road Design/Template Editor”, in the Terramodel online help.

Subgrade Template

A *subgrade template* is similar to a finish template, except that it is used to define the roadway's pavement courses, base courses, and subgrades that lie beneath the finished surface. These various courses are each referred to herein as *subgrades*.

You use the graphical **Subgrade Editor** to defined the subgrades from shapes. Again, you designate which subgrade shapes respond to superelevation, and then define and apply the transitions for the subgrade template. You can find more information about defining a road subgrade in **ROAD Menu**, "Road Design/Subgrade Editor", in the Terramodel online help.

Island Template

A *island template* is similar to a finish template, except that it is used to define the roadway's design in relationship to the centerline islands of a road design.

You use the graphical **Island Editor** to defined the island template from shapes. You can find more information about defining an island template in **ROAD Menu**, "Road Design/Island Editor", in the Terramodel online help.

Cross-sections

As indicated in the above discussion of surfaces, they can be either sliced from a DTM or stored as measured cross-section data based on offset/elevation pairs at a particular station. **Terramodel** uses cross-sections to determine where a template will intersect existing ground and to compute earthwork volumes. You can import cross-sections that have been stored in an ASCII file, or have **Terramodel** interpolate them from a DTM. If you have toggled on the Interactive design setting in Design Settings, the graphical **Xsection Editor** will show the roadway design at any cross-section defined for the road job. You can find more information about editing and viewing cross-sections in **ROAD Menu**, "Road Design/Xsection Editor", in the Terramodel online help.

Superelevation

A superelevation definition consists of the stations, lengths and cross slopes used to define the superelevation of shapes in a roadway's finish or subgrade templates. Use the superelevation editor (**Superelev** command) to define superelevations. You can find more information about superelevations in **ROAD Menu**, "Road Design/Superelevations", in the Terramodel online help.

Skip

A skip is a section of a road job where no design or volume accumulation occurs. This can be useful in modeling bridges and intersections. Use the skip manager to define and edit skips in a road job. You can find more information about defining skips in **ROAD Menu**, “Road Design/Skip manager”, in the Terramodel online help.

Roadway Segments and Station Equations

Should station equations be desired, you can associate any number of gap or overlap equations with a HAL. Since overlap station equations create circumstances where two or more positions along an alignment can have the same station, **Terramodel** uses *roadway segment numbers* to allow you to refer to each of these locations specifically. A HAL with no station equations contains 1 roadway segment which extends from beginning to end. When you assign the first station equation, the portion of the HAL before the *back* station is defined as roadway segment 1, while that portion following the *ahead* station (and preceding the back station of the next equation) is defined as segment 2. Each time you define a station equation, the HAL is resegmented, and contains a starting segment, plus an additional segment for and between each equation.

If you define an overlapping station equation as follows,

$$\text{Sta. 16+00.00 back} = \text{Sta. 13+00.00 ahead}$$

you’ve create a 300’ long section of roadway segment 2 for which station values duplicate those of segment 1. When entering the station value 15+00.00 in a station control, **Terramodel** will interpret that to be the location at 15+00.00 in segment 1 since it is encountered first. If you want to designate the location at 15+00.00 within roadway segment 2, you would enter **1500.00:2**, explicitly indicating that the desired station is within roadway segment 2. The segment number follows the station value separated by a colon. **Terramodel** will also use this convention, displaying station values and roadway segment numbers, when station equations have been defined.

Horizontal Alignment Offsets

In registering a HAL with the HAL Manager, you select a set or polyline record and register that object’s record number with the HAL. You can create a HAL that is offset from another alignment, if you wish. You can also register an alignment with multiple HALs. You can therefore create an alignment that is at the center of a divided highway, and create two HALs representing the North and South bound sections, registering the main alignment record with both HALs. In this case, the HAL’s geometry is defined by offset(s) from the main alignment.

You can specify a single offset for the entire alignment, or varying offsets at designated stations along the main alignment. Whenever offsets are used, the roadway stationing is always computed with respect to the main alignment's geometry.

Phases

You can use *phases* to distinguish between the individual portions of a roadway cross-section that are constructed independently in order to facilitate the maintenance of traffic as existing facilities are widened. You create named phases and then define, within the graphical **Phase Editor**, the individual regions of the cross-section that relate to each phase. Volumetric and Mass haul analyses can be performed distinguishing between the work to be done in each phase. For more information about phases, see **ROAD Menu**, "Phase Names", and "Mass Haul/Phase Editor", in the Terramodel online help.

The Vertical Alignment Editor

The **Roadway Vertical Alignment Editor** dramatically simplifies and speeds the process of editing a vertical alignment and fitting it to an existing ground profile. User definable design criteria allows you designate the geometric constraints that apply to a vertical alignment or to load this data from predefined design criteria sets. You'll be able to implement the commonly used AASHTO geometric criteria or define other applicable design constraints and considerations. The process of editing the vertical alignment employs a user interface of remarkable simplicity, where the desired actions are in some cases simply inferred based on where you snap on the alignment. You simply drag the alignment into the shape that you want. As a smart real-time visual editing solution, the vertical alignment editor handles the imposition of the design criteria, by not allowing you to drag the alignment into a configuration that violates the current criteria. The alignment's geometry can be developed purely visually, with full confidence that whatever alignment geometry you can achieve is in fact acceptable. Alternatively, you can enter specific geometric data as you wish, either fully defining or further constraining the solution. For more information about editing a vertical alignment, see **ROAD Menu**, "Alignments/Roadway vertical Alignment Editor", in the on-line help system.

Mass Haul Diagrams

Terramodel can generate a mass haul diagram illustrating the over all mass ordinate and the accumulative mass data for each material and phase. You can find more information about the creation and analysis of mass haul diagrams in **ROAD Menu**, "Mass Haul/Diagram", in the Terramodel online help.

Other Uses

The roadway design module, while focusing on roadways, is in no way limited to the design of roadways alone. It is commonly used for channel dredging, canal, dam, levee, or other earth embankment or excavation design purposes.

Creating a Roadway Design

Create your materials.

A road must be built in and of a material, and so you must define what materials will be used in the design (i.e., Fill, Cut, Dirt, Asphalt, Concrete, etc.). At a minimum, a fill material must be defined, and is defined automatically by **Terramodel** the first time you enter the material manager. If you have performed a subsurface investigation in order to model the various soil strata that will be encountered, you'll wish to define materials for each of those (i.e., Clayey Fine Sand, Sandy Clay, Muck, Weathered Rock, etc.).

Create surfaces for the design.

At a minimum the existing ground surface must be created, either as a DTM or as stored cross-section data. The order of the surfaces is important in helping **Terramodel** to properly model the design and determine material quantities. (The finish surface must be defined in order to view the interactive design.)

Create the shape classes.

Shape classes are listed and organized in a shape library. Use shape classes to define the categories (ditches, roadbeds, shoulders, etc.) in which shapes are organized.

Create your shapes.

Shapes are the building blocks for the construction of templates. Templates consist of collections of shapes, so any feature that you want in a template must be defined as a shape. Shapes have unique properties which, when used in a template, control how the template behaves in any given situation.

Create your horizontal alignment(s).

Use the **Pline** or **Set** command and the Curve option of **Edit Object** to create a horizontal alignment.

Register HALs in the HAL Manager.

Once you've created your HAL, add it to the HAL Manager as a New Alignment.

Define station equations and horizontal offsets.

While in the HAL Manager, define any station equations for your horizontal alignment. You can also modify HALs using offset distances from the specified alignment.

Store vertical alignment(s).

Let **Terramodel** interpolate and draw the existing ground surface profile in the profile view from the existing ground DTM using the **Profile** command. Use the **Pline** command and the **RDValEdit** command to design the roadway's vertical profile.

Register VALs in the VAL Manager.

Add your vertical alignment to the VAL Manager as a New Alignment.

Define vertical offsets.

Also from within the VAL Manager, you can modify the vertical profiles using offsets from the specified vertical alignment.

There is a Terramodel Training Guide module dedicated to completing a roadway design. See this guide for additional details.

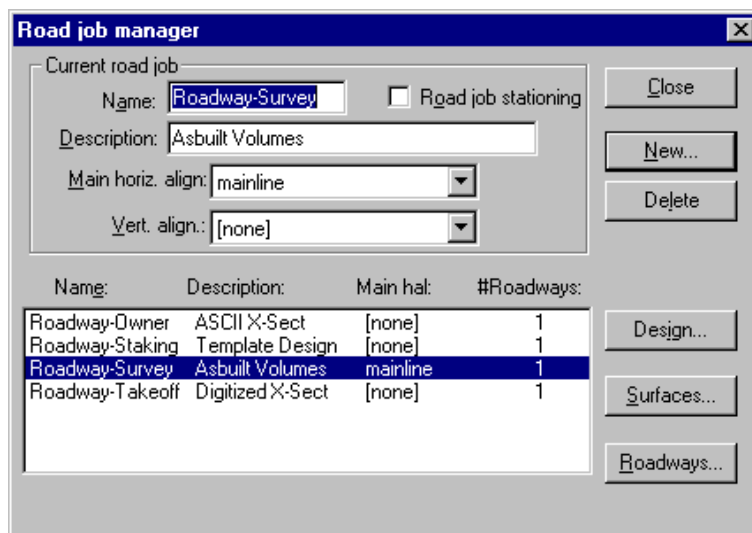
Using Road Job Stationing (RoadJob)

Road job stationing is a feature that allows you to relate locations along any HAL in the project, Left_Curb and Right_Curb for instance, to the stationing along the designated centerline HAL of the road job. This can simplify creating HAL/VAL and HAL/Slope transitions.

Note: You should decide at the beginning of the project, when you create the road job, whether to enable road job stationing.

Enabling Road job stationing

1. Select the Road design/Road job manager option from the Roads menu. The *Road job manager* dialog box appears.



2. To enable the road job stationing, click the Road job stationing checkbox and verify the name of the HAL that you want to use as the job stationing reference in the *Main horiz. align* field.

Note: Do not apply the road job stationing to existing Road jobs created in previous versions of Terramodel. If the Road job was working properly in the previous version and you enable this checkbox, Terramodel may use incorrect station references for some transitions.

Designing a Single Roadway

Create the road job.

Using the Road Manager, store a road job, assigning a name and associating it with a main horizontal alignment. As you store a new road job, **Terramodel** automatically creates a default roadway with the same horizontal alignment. The Road Manager uses the vertical alignment only in association with the roadway (and not the road job) but lists it with the road job for convenience in designing simple road jobs involving only one roadway.

Establish the roadway data

The default roadway that **Terramodel** created when creating the road job has the same name as the road job. You can now rename it if you like, and establish its vertical alignment. If you wish to use the roadway vertical alignment editor to edit the vertical alignment object, associated with this roadway, you'll first have to establish the roadway design criteria, by clicking on the Design button in the Roadway Manager.

Create or import cross-sections.

Use the Import feature in the Xsection Manager to import previously defined cross-sections for use in your road design, or have them sliced from a DTM. You can also manually enter a cross-section's offset and elevation data in the Xsection Editor. Use the Interactive Design feature in Design Settings to see how your design is applied at each cross-section.

Define your templates.

Using the Template Manager and Template Editor, define the beginning stations and geometry for each template in the roadway. You can choose to define a few templates now, then finish incomplete templates and/or add more later. Use the Interactive Design feature in Design Settings to see how your template is applied to the cross-section at its initial station.

Define your subgrades.

Using the Subgrade Manager and Subgrade Editor, define the beginning stations and geometry for each subgrade in the roadway. You can choose to define a few subgrades now, then finish incomplete subgrades and/or add more later. Use the Interactive Design feature in Design Settings to see how your subgrade is applied to the cross-section at its initial station.

Define phase lines if phased construction is needed

Using the Subgrade Manager and Subgrade Editor, define the boundaries of the individual phase limits.

Calculate earthwork quantities and print other reports.

Use the **Earthwrk** command to generate reports including borrow and waste material and closed shape quantities. Use the **RoadRpt** and **Staking** commands to obtain alignment, template, tie and staking information.

Plot cross-sections.

The **RDX** command (**Plots/Cross-sections** selection on the menu) or the **Planset** command are each able to create finished cross-section sheets. The sheets can then be plotted using the standard **Terramodel** plot features.

Designing Multiple Roadways

Create the road job.

Using the Road Manager, store a road job, assigning a name and associating it with a main horizontal alignment. As you store a new road job, **Terramodel** automatically creates a default roadway with the same horizontal alignment. The Road Manager uses the vertical alignment only in association with the roadway (and not the road job) but lists it with the road job for convenience in designing simple road jobs involving only one roadway. Note that the road job that you create or select will become the *CURRENT ROAD JOB*.

Create the roadways.

Define each of the roadways that will be included within the road job, specifying the registered HAL and VAL for each. The roadways that belong to the same road job will be stationed based on the same main alignment, as specified for the road job when it was created. These roadways are also expected to be intersected by the same xlines for use by the Xsection manager. Use the **Road Design/Roadways/Roadway Manager** menu selection to create, delete, and edit roadways. Note that the roadway you select will become the *CURRENT ROADWAY*. If you wish to use the roadway vertical alignment editor to edit the vertical alignment object, associated with these roadways, you'll first have to establish the roadway design criteria for each, by clicking on the Design button in the Roadway Manager.

Create or import cross-sections.

Use the Import feature in the Xsection Manager to import previously defined cross-sections for use in your road design, or have them sliced from a DTM. You can also manually enter a cross-section's offset and elevation data in the Xsection Editor. Use the Interactive Design feature in Design Settings to see how your design is applied at each cross-section.

Define your templates for each roadway.

Using the Template Manager and Template Editor, define the beginning stations and geometry for each template in each roadway. You can choose to define a few templates now, then finish incomplete templates and/or add more later. Use the Interactive Design feature in Design Settings to see how your template is applied to the cross-section at its initial station. If Interactive Design is toggled on in Design Settings, you will also see the template designs for other roadways in the road job.

Define your subgrades.

Using the Subgrade Manager and Subgrade Editor, define the beginning stations and geometry for each subgrade in each roadway. You can choose to define a few subgrades now, then finish incomplete subgrades and/or add more later. Use the Interactive Design feature in Design Settings to see how your subgrade is applied to the cross-section at its initial station. If Interactive Design is toggled on in Design Settings, you will also see the subgrade designs for other roadways in the road job.

Define phase lines if phased construction is needed

Using the Subgrade Manager and Subgrade Editor, define the boundaries of the individual phase limits.

Calculate earthwork quantities and print other reports.

Use the **Earthwrk** command to generate reports including borrow and waste material and closed shape quantities. Use the **RoadRpt** and **Staking** commands to obtain alignment, template, tie and staking information for each roadway.

Plot cross-sections.

The **RDX** command (**Plots/Cross-sections** selection on the menu) or the **Planset** command are each able to create finished cross-section sheets. The sheets can then be plotted using the standard **Terramodel** plot features.

Mass Haul Analysis

A careful mass haul analysis helps you critique your road job operation, so you can streamline haul routes and productivity. Terramodel has the tools you need for making a careful mass haul analysis. You can use the road job you already have set up in Terramodel, so you do not need to re-input the job.

Preparing a Project to do a Mass Haul Analysis

Prepare the project as you would any other: import or design your road job data. When you are ready to begin your mass haul analysis, make sure you take the following steps:

- Open the project you want to work with.
- Define the materials available at the job site.
- Set up the phases if you plan to work in phases.
- Assign any significant points (e.g. roadways, bridges, etc.) that will affect your diagram.
- Set up the subgrade materials in the Subgrade Editor if you plan to balance your materials in the mass haul diagram yourself.
- Set up the mass hauling properties for the materials, including marking subgrade materials as Embankment.

Defining Materials Available at the Job Site

Use the *Material Manager* to create a list of all the materials (such as asphalt, concrete, sand, clay, gravel, rock, muck, topsoil, etc.) you might need to factor into your road design and to assign certain properties to those materials. The Roads module uses the materials you define here to:

- Perform separate volume accumulations for each type of material. (Volumes are accumulated only for the defined materials.)
- Designate the composition of roadway pavements and subgrades.

You can also use this list of materials in other dialog boxes to assign a material type to a subsurface soil strata, a roadway subgrade, or a shape.

Each Terramodel project or prototype file automatically has the minimum number materials predefined: *Cut* and *Fill*. You can rename these materials and add as many other materials as you need. If you place all your earth materials into the default fill or cut materials, Terramodel cannot distinguish amongst them to give you specific volumes the materials available at the job site.

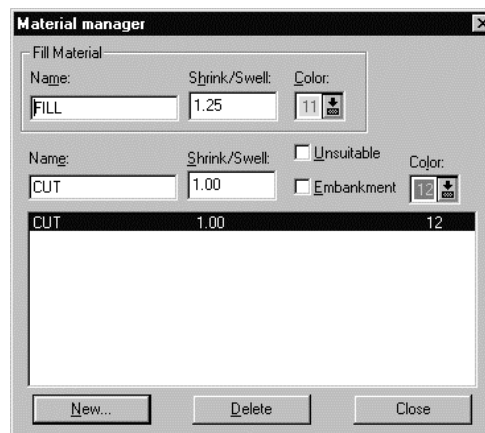
Each surface or subsurface strata you create will be composed of a designated material, allowing ROADS to determine quantities of cut and fill based on material types. In

addition, closed shapes can have a material type as a property, so ROADS can determine the quantities of pavement construction materials.

To use the Materials Manager:

1. Type **Materials** into the command bar.
OR
Click the **ROAD** menu, and click **Materials**.

The *Material Manager* dialog box appears.



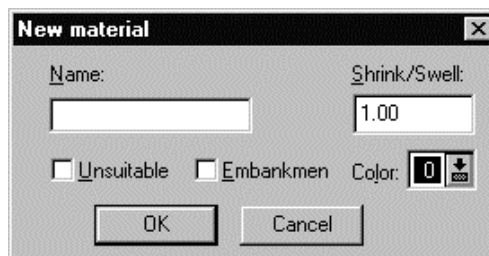
2. Change the properties of the default fill material in the Fill Material section of this dialog box.

You cannot delete this material or mark it as unsuitable or as embankment. Change the default fill material properties as desired.

3. Click **Close** when you are done editing your materials.

To create a new material:

1. Click **New**. The *New Material* dialog box appears.
This dialog box lets you define a new material, which can then be assigned to a surface, a subgrade, or to a closed shape. The material is added to the *Material Manager* dialog box.



2. Type a name into the Name box. Press **Tab**. The focus moves to the Shrink/Swell edit box.
3. Type in the shrink/swell factor that represents what happens to this material when it is used as fill material.
4. The shrink/swell factor represents the difference between the original in-place volume of the currently highlighted material before it is cut, and its final in-place volume after being placed and compacted into fill.

Swell numbers are positive numbers greater than 1 and less than 2. For example, 1.2 equals a 20% swell.

Shrink numbers are positive numbers less than 1. For example, .8 equals a 20% shrink.

5. If the material is unsuitable to be included in the road design, click the Unsuitable check box to place a check mark in it.
6. Unsuitable materials cut by a surface or subgrade shape in the template are completely removed. Unsuitable materials, so removed, are not available for use in fill. A typical use for this option is topsoil stripping.
7. If the material is suitable for embankment material, click the Embankment check box to place a check.

Embankment material must be suitable as subgrade material for the kind of road you are building.

8. Select a new color for the material, if desired.
9. Click **OK** to add the material to the Materials Manager.

To change a material:

1. Click the material. Its properties appear in the boxes above the list of materials.
2. Change the properties as desired.

To delete a material

- Click the material, then click **Delete**.

Setup Phases for the Road Job

Mass Haul/Phase Editor

The **Phase** command displays the Phase Editor where you define the cross-sectional extents of the various phase limits within a phase template.

Mass Haul/Phase Manager

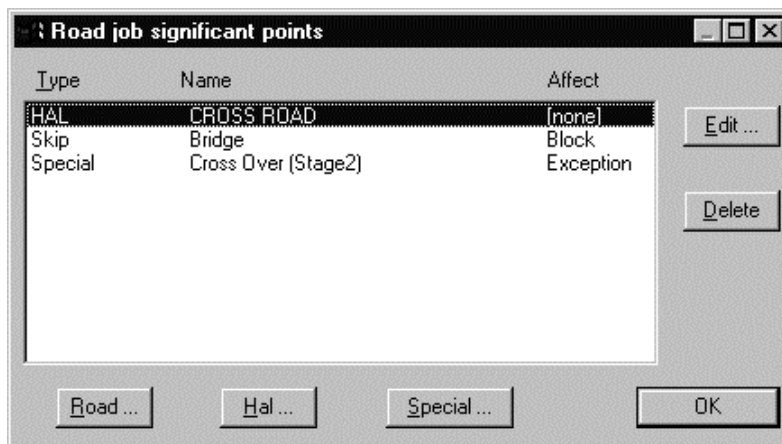
The **PhaseMan** command opens the Phase Manager dialog box where you can define the stationing limits and the cross-sectional extents of individual phases. Use the **PhaseMan** command, for example, to design a roadway to be constructed in two or more phases to ease traffic maintenance. Volumes and mass haul information can then be reported on a phase by phase basis.

Set up Your Significant Points

The Significant Points dialog box lets you designate where significant points such as roadways, bridges, or other site features cross the road job's main alignment. Use significant points to note places along the road job where hauling is in some way affected by the presence of some obstacle that may delay or block hauling. You can mark a significant point to have no effect on hauling, to block the haul route, or to calculate the hauled material separately. Skips always affect the mass haul diagram and will automatically be labeled.

To work with significant points:

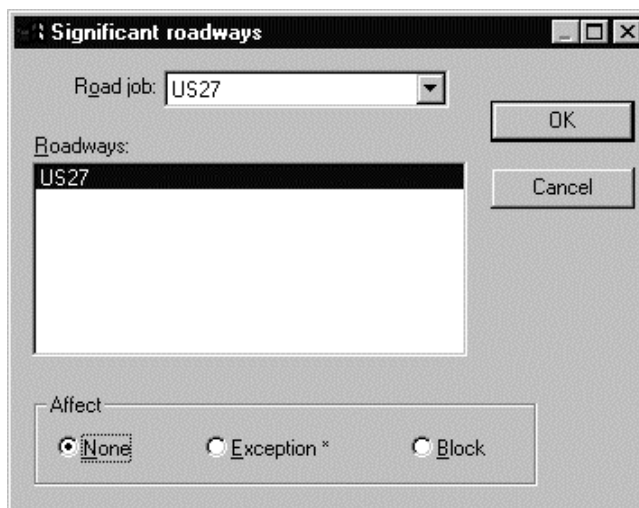
1. Type **RoadSigPoint** into the command bar.
OR
Click ROAD, Mass Haul, and Significant Points.
The *Road Job Significant Points* dialog box appears.



2. Mark and edit your significant points.
3. Click **OK** when you are done.

To mark a significant roadway:

1. Click **Road** in the *Significant Points* dialog box.
The Significant Roadways dialog box appears. This dialog box allows you to designate an intersecting roadway as being significant in the analysis of a mass haul diagram, and to indicate the affect it has on your diagram.

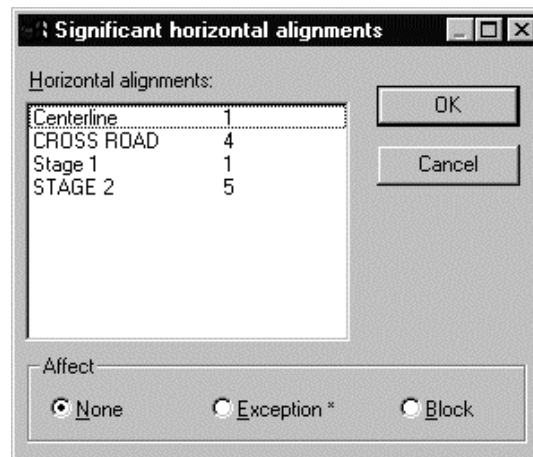


2. Click the Road Job list box, and click on the road job you want to select.
3. Click on the roadway you want to mark.

4. Click an affect option button.
5. Click **OK**.

To mark a significant HAL:

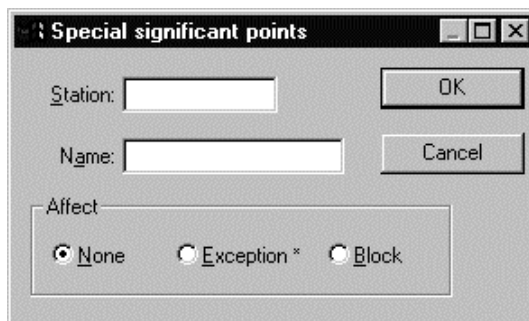
1. Click **HAL** in the *Road Job Significant Points* dialog box.
The *Significant Horizontal Alignment* dialog box appears.



1. Click on the HAL you want to mark.
2. Click an Affect to apply to the HAL.
3. Click **OK**.

To mark a special significant point:

1. Click **Special** in the *Road Job Significant Points* dialog box.
The *Special Significant Points* dialog box appears. This dialog box allows you to designate the location of any particular feature or situation that may have an impact on the hauling of fill material and therefore on the balancing of the mass haul diagram.



1. Use the Station control to enter the station you want to mark as significant.
2. Type the name of the significant point, if desired.
3. Click an affect to apply to the significant point.
4. Click **OK**.

To edit a significant point

1. Click on the point, and click **Edit**. A dialog box appears.
2. Modify the information in the dialog box.
3. Click **OK**.

To delete a significant point

- Click on the point, and click **Delete**.

Establishing Subgrades

A subgrade represents a particular material placed beneath the finish surface. Subgrades are typically used to model pavement courses, base courses, stabilized subgrades, etc., each one being defined as a separate subgrade. Subgrades are assigned levels indicating the order in which they were created. The level number is just a designation used to denote where a new subgrade is started. It identifies the subgrade's vertical position, level 1 typically being the uppermost subgrade, i.e. immediately beneath the finish surface. Each subgrade is assigned a unique level number L-1, L-2, L-3, etc.

A subgrade is defined by one or more shapes, and has an associated material type and a depth. In the case of the uppermost subgrade, its depth is measured from the finish template surface. The depth of subsequent subgrades is measured from the bottom of the subgrade beneath which they lie.

It is important to understand that subgrades are intended to be very flexible, giving you the ability to place a subgrade anywhere it is needed. The beginning of a subgrade is located horizontally by a combination of the Locate by, Horiz. Align., Chain, and Offset controls. The vertical position is controlled by the Locate by, Depth, and Chain controls. Using these combinations should allow subgrades to be placed in virtually any location desired.

To use the subgrade editor:

1. Type **Subgrade** into the command bar.
OR
Click **ROAD**, **Road Design**, and **Subgrade Editor**.
The Subgrade Editor appears.
2. Set up your subgrades.
The following table describes what each menu does for you.

Subgrade Editor Menus

File	This menu contains commands that let you create and save subgrade templates.
Display	This menu contains commands that let you control how the template is displayed in the Subgrade Editor graphics screen.
Transition	This menu contains commands that let you apply transitions to either the left or right shape currently highlighted in the Subgrade Template Shapes list box.
Shape...	This option opens the Shape Editor graphics screen, where you can create new shapes.
Settings	This menu allows you to open the Road Design Settings dialog box, where you can establish the design settings for the roadway, or the Editor Grid Settings dialog box, where you can alter the way the Template Editor grid is displayed and labeled.
Warnings...	This menu choice will be enabled only when certain questionable design conditions occur. Selecting this menu choice opens the Road Warnings dialog box, where you can review the problem.

The controls within the Subgrade Editor let you assign beginning stations to subgrade templates and select subgrade shape classes and shapes that will comprise the template.

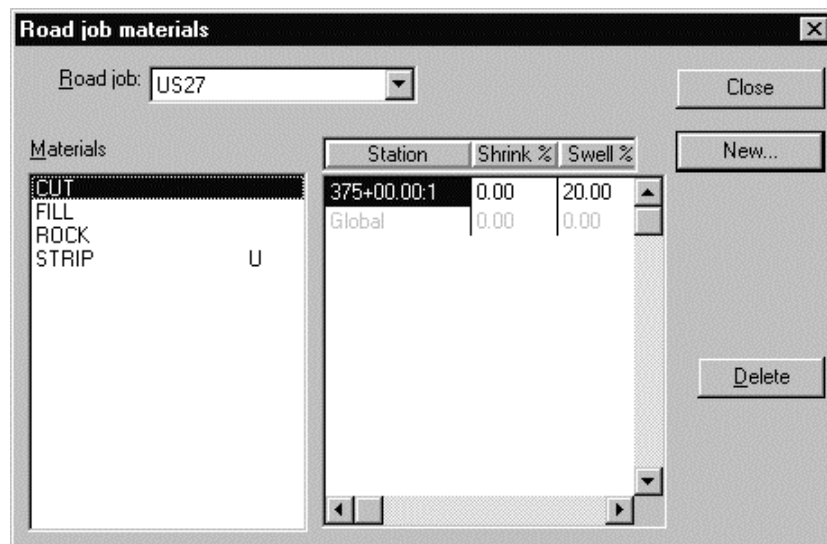
Define Hauling Properties for Road Job Materials

The Road Job Materials dialog box lets you define the hauling volume properties for each material. These properties include the shrinking and swelling percentages for materials that define their loose volumes as they are being trucked or otherwise hauled, as compared to their in-place volume. You can also indicate whether or not these materials are acceptable for use within embankment, and you can limit the acceptable composition of fill materials. You can refine the designation of these hauling volume properties to reflect changing conditions along the course of a particular road job. These settings are used only by the mass haul computations.

To define hauling volume properties:

1. Type **RoadMat** into the command bar.
OR
Click **Road**, Mass Haul, and Road Job Materials.
The Road Job Materials Dialog box appears. You created the materials listed in this dialog box when you set up the materials for your project.

Click **Close** when you are done working with road job materials.



To select a different road job:

- ▶ Click the **Road Job** box, and click on your selection.

To add a new hauling property to a material:

1. Click on a material in the Materials list box.
The material's hauling properties appear in the Station/Shrink%/Swell % list box.
2. Click **New**. The *Hauling Volume Properties* command bar appears.

The *Hauling Volume Properties* command bar lets you define new hauling volume properties for a material you select in the *Road Job Materials* dialog box beginning at a station you select. The hauling properties you select in this command bar remain in affect until you define other hauling properties at a different station.

Station: <input type="text"/>	Shrink: <input type="text" value="0.00"/>	Swell: <input type="text" value="0.00"/>	<input type="button" value="OK"/>	<input type="button" value="Cancel"/>
RoadMat	Create factors for a road job material.			Station

3. Use the Station control to select a station where you want the new hauling properties to begin.
4. Use the Shrink % real number control to enter a shrink percent, if desired.
5. Use the Swell % real number control to enter a swell percent, if desired.
6. Click **OK**. The command closes and takes you back to the *Road Job Materials* dialog box.

To delete a material or a hauling property for that material

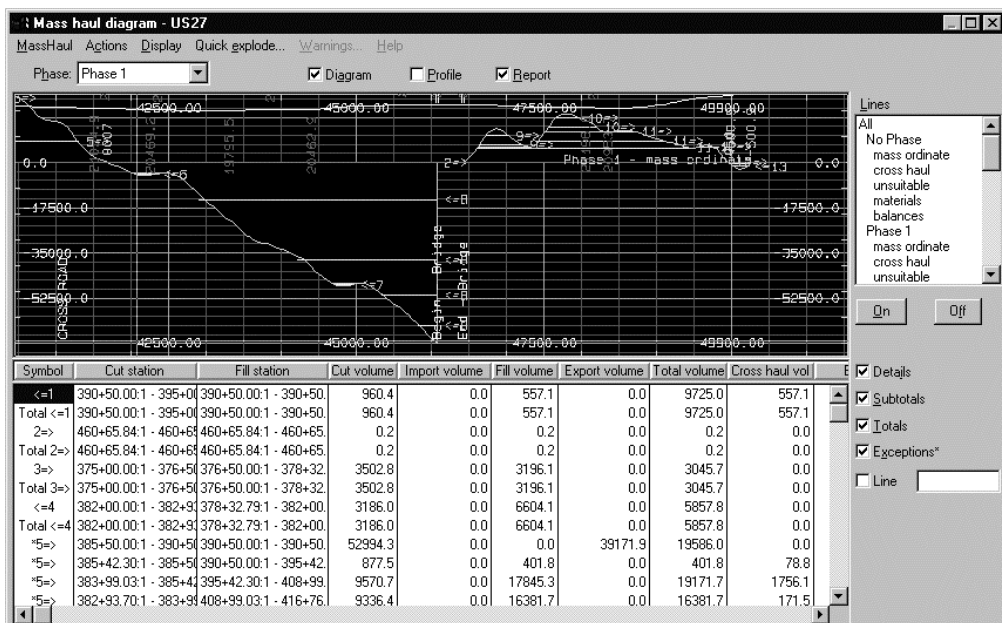
- ▶ Click on a material, and click **Delete**.

Using the Mass Haul Diagram

The mass haul diagram is a power tool for helping you find the best haul routes for any phase of the job. You can see the mass haul information graphically in diagram and profile views and in a text report. You can set up and select phases, set haul ranges, define significant points, specify material for import or export, and balance areas along the job. Terramodel can balance your materials for you or you can balance materials individually for more control over your mass haul analysis. The Mass Haul Diagram report shows one material report at a time and automatically updates as you work with your mass haul diagram.

To use the mass haul diagram:

1. Type **MassHaul** into the command bar.
OR
Click ROAD, Mass Haul, and Diagram.
The *Mass Haul Diagram* appears.



As the mass haul diagram for the current road job is created, the main horizontal alignment for that road job is automatically set as the active alignment and appears as a white line in the diagram. The white line represents your working mass ordinate (the net of cuts and fills for a particular material).

If you make changes to the working mass ordinate, the new data will assume the white color and the original data will be displayed in a different color.

The lines in the diagram represent the cut and the fill at a given point along the alignment. "Hills" in the diagram represent areas of cut while "valleys" in the diagram represent areas of fill.

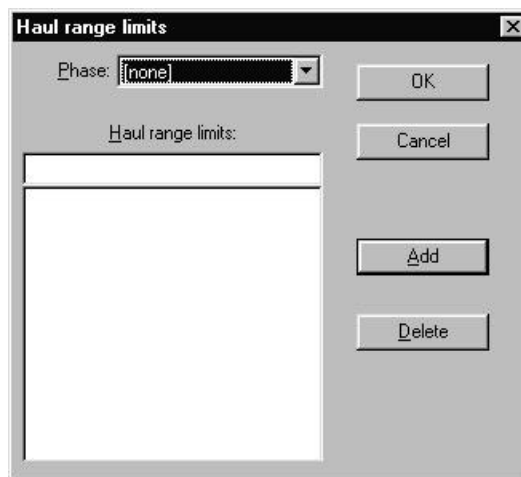
2. Click the **Diagram**, **Profile**, or **Report** check box to select or deselect a view.
3. Set up your haul range limits. See "Setting up Haul Range Limits."
4. Select the phase you want to work with by clicking on the phase box, and clicking on the phase.
5. Mark positions for material import or export.
6. Balance your materials. See "Balancing Materials."
7. Send the *Mass Haul Diagram* report to the report generator, by clicking on **Actions**, and then clicking on **Reports**.
8. Click **FILE** and **Exit** to close the *Mass Haul Diagram*.

Setting up Haul Range Limits

The Haul Range Limits dialog box lets you set up the hauling distances for the project for your Mass Haul Diagram. The Mass Haul Diagram will calculate mass haul data for each range you set up in this dialog box, so you can better determine the costs for hauling material.

To set up your haul range limits:

1. Click **Actions** in the *Mass Haul Diagram*.
2. Click **Haul Ranges**. The *Haul Range Limits* dialog box appears.
3. If your project has been divided into phases, click the **Phase** box, and click on the phase you are currently diagramming.



To add a haul range:

1. Type a haul range into the small *Haul Range Limits* box.
2. Click **Add**. The range is added to the list of haul ranges in the large Haul Range Limits box.
3. Click **OK** when you are done setting up your haul range limits.

To delete a haul range

- Click on the range in the large Haul Range Limits box, and click **Delete**.

Balancing materials

To auto-balance the entire phase:

1. If "All" is not selected in the Material box, click the **Material** box, and click **All**.
2. Click **ACTIONS** and **Auto-Balance**.

To balance each material individually:

1. Click the Material box, and click on the material you want to balance.
2. Click **Actions** and **Balance by Range**.
3. Click on the first range in the diagram you want to balance. You can click on either a "hill" or a "valley" in the diagram.

Balance each range in Terramodel the way you plan to balance each range on the site. Mass Haul volumes are affected by the order in which you balance the ranges.

Lines appear within the range, once it has been balanced, showing the amount of material for each range limit you have set up. A symbol denoting the general direction of dirt movement along with the balance number appears on the diagram.

Sometimes a message box appears telling you that Terramodel is not able to balance at the selected point. This message box generally appears when the material is already balanced or when you have not set up haul ranges.

4. Repeat step 3 until you are done balancing the material.

To remove all the balances from your diagram

- Click **ACTIONS**, **Balance**, and **Clear All Balances**.

To remove one balance at a time

- Click **ACTIONS**, **Balance**, and **Delete Balance**.

Mass Haul Diagram/Actions Menu

The Mass Haul Diagram has an Actions menu that lets you work with phase names, haul ranges, significant points, import/export, balancing haul ranges, and modifying your text reports. To use any of the features described below, click **ACTIONS** in the *Mass Haul Diagram* dialog box, then click on the command you want to run.

Phase Names

This command opens the *Phase Names* dialog box, where you can create a new phase, delete an existing phase, and alter an existing phase's settings.

Haul Ranges

This command opens the *Haul Range Limits* dialog box, where you can define haul ranges.

Sig Points

This command opens the *Road Job Significant Points* dialog box, where you can designate that the station at which the current road job's main alignment crosses certain significant roadways, alignments or other site features be noted on the mass haul diagram. A significant point might be a location along the road job at which the hauling is in some way affected by the presence of some obstacle causing a blockage or delay.

Import/Export

This command opens the *Material Import/Export* dialog box, where you can designate that a specified volume of a certain material be imported or exported at a particular station along the mass haul diagram.

Balance

This command lets you balance the material you plan to haul. You can:

- Let Terramodel balance the material along the site for you with *Auto-Balance*.
- Balance an individual haul range with *Balance Haul Range*.
- Remove all existing balance information with *Clear All Balances*.
- Delete the balance information from an existing haul range with *Delete Balance*.

Report Labels

This command opens the Report Column Headings dialog box where you can select what information is included in your report and you can order the information to suit your needs.

Reports

This command sends the Mass Haul Diagram to the report editor.

Mass Haul Diagram/Display Menu

The vertical exaggeration of the profile view graphics screen within the mass haul diagram is controlled by that assigned to the profile viewing mode. See SETTINGS Menu, View Settings in the online Help system for information on setting the vertical exaggeration of a viewing mode.

All

This command displays the entire Mass haul diagram and vertical alignment profile in their respective graphics screens. The shortcut key combination Ctrl+A will execute this command while in the *Mass Haul Diagram* dialog box.

Redraw

This command refreshes both graphics screens. The shortcut key combination Ctrl+R will execute this command while in the *Mass Haul Diagram* dialog box.

Magnify

This command magnifies the graphics displayed within graphics screens using a magnification factor of 0.85, thereby zooming the image out by a fixed amount. The shortcut key combination Ctrl+M will execute this command while in the *Mass Haul Diagram* dialog box.

Pan

This command shifts the image in the indicated graphics screen from one position to another. You specify the *From* and *To* positions by picking them with your mouse. The shortcut key combination Ctrl+P will execute this command while in the *Mass Haul Diagram* dialog box.

Zoom

This command resizes a specified area of either graphics screen to a larger window scale. You specify the resize area by picking the corners with your mouse. The shortcut key combination Ctrl+Z will execute this command while in the *Mass Haul Diagram* dialog box.

Recenter

This command pans the image to place the designated position at the center of the screen. You designated the position to be recentered by picking it with your mouse. The shortcut key combination Ctrl+E will execute this command while in the *Mass Haul Diagram* dialog box.

Previous

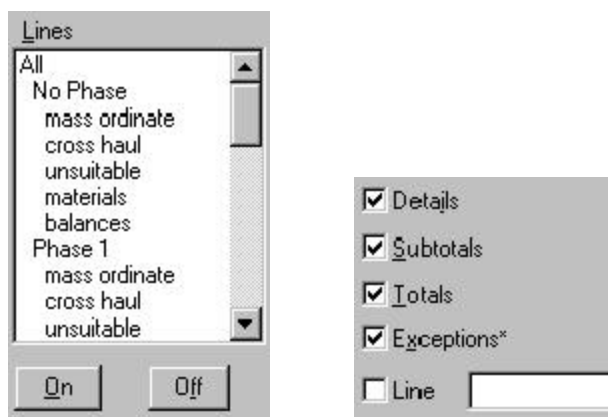
This command returns the image to its previous state, after having performed one of the above image resizing commands. The shortcut key combination Ctrl+U will execute this command while in the *Mass Haul Diagram* dialog box.

Options

This command lets you see the display options for the Mass Haul diagram and the Mass Haul report. The options for the diagram let you turn lines in the diagram on and off. The options for the report let you show or hide portions of the report, so the report can be as simple or as detailed as necessary. The diagram must be visible for the diagram options to appear. The report must be visible for the report options to appear.

To see the display options:

1. Click **Display** in the Mass Haul Diagram dialog box.
2. Click **Options**. The options for the diagram and/or report appear. The first graphic below shows the options that appear for the diagram. The second graphic below shows the options that appear for the report.



3. To turn a line in the diagram on or off, click on a line in the Lines box, and click On or Off. If you select All and turn it on, all the available lines appear in the diagram.
4. To show or hide report details, subtotals, totals, or exceptions, click on a check box. These options are rows in the report.
5. To see the raw data for a material line, click Line, and click on the material line in the diagram.
6. To hide the options, repeat steps 1 and 2.

Settings

This command presents the Settings sub-menu allowing you to open the *Road Design Settings* dialog box, or the *Mass Haul Diagram Grid Settings* dialog box.

Mass Haul Diagram Grid Settings

The **Scalegrid** command opens the *Mass Haul Diagram Grid Settings* dialog box, where you establish the settings for the grid displayed in the Mass Haul Diagram.

Grid Type

This section allows you to control the way in which the grid is displayed.

Dots/Ticks/Lines These radio buttons display gridlines as dots, tick marks at the grid edge, or solid lines.

Display

This section contains controls that let you designate the color in which certain items in the diagram are displayed.

- Base line** This check box/color control combination lets you designate whether or not the mass haul diagram's baseline will be drawn, and if so, its color. The diagram baseline is a horizontal line indicating the line of zero accumulated cut/fill.
- Labels** This check box/color control combination lets you designate whether or not the mass haul diagram's labels will be drawn, and if so the color used to draw the axis labels and the significant point labels. The labels associated with each individual line in the mass haul diagram are drawn using the color of that line. These include the identification labeling of the mass ordinate line and the individual phase lines, and the local minimum and maximum values of each.
- Axis** This check box/color control combination lets you designate whether or not the mass haul diagram's grid axis will be drawn, and if so, its color.
- Grid** This check box lets you designate whether or not the mass haul diagram's grid will be drawn. The following four colors are associated with the grid.
- Horz major** The color of the major horizontal grid lines is specified in this control.
- Horz minor** The color of the minor horizontal grid lines is specified in this control.
- Vert major** This control designates the color of the major vertical grid lines.
- Vert minor** This control designates the color of the minor vertical grid lines.

Gridline Spacing Interval

- Horz** This real number control allows you to designate the interval along the vertical axis at which minor horizontal grid lines will be drawn.
- Vert** This real number control allows you to designate the interval along the horizontal axis at which minor vertical grid lines will be drawn.

Major Grid Lines Every nth Interval

- Horz** This integer control designates that a major horizontal grid line be draw at each designated multiple of the minor horizontal grid line spacing interval.
- Vert** This integer control designates that a major vertical grid line be draw at each designated multiple of the minor vertical grid line spacing interval.

Label Height (pixels)

This integer control allows you to designate the label height in units of pixels.

Scaling Factor

This real number control enables you to control the vertical scale of the diagram with respect to the horizontal scale. In a mass haul diagram, the vertical axis coordinates are measured in units of cubic project units, i.e., cubic feet, or cubic meters, while the horizontal axis coordinates are in units of stationing. This factor lets you control the relative scaling of the y axis with respect to the x axis. Common values might range from 20 to 100, depending on the length of the project. If not sure about the required scale, enter a value of zero and **Terramodel** will attempt to find and set an appropriate scale factor.

Import Cut/Fill Volumes to use in MassHaul Analysis (MhImport)

MhImport, a keyboard-only command, imports a text file with cut and fill volumes. The command requires a pre-existing road job name. The road job can be empty. The imported data is stored in the **Terramodel** database. Once the data has been imported into **Terramodel**, the **MassHaul** command can perform a masshaul analysis. Currently, the only way to delete this data is to remove the road job.

To Use MhImport

Enter **MhImport** at the **Terramodel** command line. The command line displays the following fields:

MhImport command line options

Road	Select a Road job from the Road list. This control lists the current road jobs registered with the Road design/Road job manager. Terramodel must have a registered Road job before you run the MhImport command.
Phase	If phases for this road job have been registered in the Mass Haul/Phase manager, this control lists the available phases. Select one.
Import	Click Import to browse for a text file to import. This file must contain data in the format: Station, Incremental Cut Volume, Incremental Fill Volume. The fields can be tab, comma, or space delimited. Section designations in the Station number are not supported. By default, MhImport reads Excel comma delimited output files (*.csv). Once the file is selected, the import is executed.
Cancel	Click Cancel to exit the MhImport command line without executing the import.

How to Use the Pipe Commands

This section describes the general concepts of how you can use the *Sewer Design and Analysis* module (Sewer for short) to design and analyze sewer systems. Most of the commands for this module are in the **PIPE** menu and are secured to this module.

Drainage Concepts and Terminology

Terramodel designs sewers (sanitary or storm drainage systems) based on a pipe and node system. “Nodes” can represent manholes (inlets are included in this category), headwalls, junction boxes, and abrupt transitions (pipe size, material, or grade changes). A “pipe” is a broad term that **Terramodel** applies to both pipes and open channels. The entire system of pipes and nodes is called a “branch”.

Like the **HYDROLOGY** module, the **SEWER** module utilizes object attributes. **Terramodel** assigns your input parameters and the results of hydraulic computations to **Terramodel** points (nodes) or sets (pipes) as attributes. Once an attribute is defined, you can recall the information and even use it in other types of calculations.

Nodes

A node is represented in **Terramodel** by a point object. **Terramodel** computes the hydraulics at each node and assigns them to the point as attributes. Your selection of node shapes and sizes is contained in the file **TMNODE.SIZ** (**TMNODE_M.SIZ** for metric sizes), and Manning’s coefficients for the available shapes are contained in the **TMNODE.TBL** file, both in the **Terramodel** directory. You can edit these files to customize node shapes, if necessary. Depending on the type of node you specify, **Terramodel** draws each node type differently using settings you enter in the **Pipedef** and **Node** commands. As with all TMLs, you can edit the **PipeDraw** TML command to customize the node graphics to your standards.

All node types except for transitions consider the inlet, outlet and node loss coefficients. A transition node is a special type of node that you can also use to model the situation where there is no free opening to the atmosphere (or no structure), or when you want more control over the headloss calculations at the junction. When you specify a node as a transition, **Terramodel** ignores pipe inlet and outlet losses and considers only the node loss coefficient.

You can enter flow (**Q**) values directly at each node, or **Terramodel** can calculate the flows from contributing basin information that you supply.

Pipes

A pipe is represented in **Terramodel** by a set object. You establish the direction of flow through a pipe by entering the set starting at the downstream end and working upstream. After you've entered the set, you can assign pipe properties to it. The pipe properties that you enter and properties that **Terramodel** calculates are assigned to the set as attributes. **Terramodel** uses this attribute information to draw the pipe, and updates it each time a property changes or the system is re-calculated. As with nodes, you can change how the pipes are drawn by modifying the **PipeDraw** TML command.

Your selection of pipe shapes and sizes is contained in the file **TMPIPE.SIZ** (**TMPIPE_M.SIZ** for metric sizes), and Manning's coefficients for the available shapes are contained in the **TMPIPE.TBL** file, both in the **Terramodel** directory. You can edit these files to customize the pipe shapes and sizes, if necessary.

Branches

A branch is a pipe-node combination. The discharge end of a branch should always be the first point on the set so that **Terramodel** can calculate water surfaces starting from the downstream end (**Terramodel** also stations branches in this direction).

Pipe Systems

A pipe system is a collection of one or more branches. **Terramodel** has the ability to analyze multiple branches simultaneously, determining the hydraulic relationship between the branches. You aren't required to connect branches together, but **Terramodel** won't know that they connect unless the common node is represented by the same point object.

Flow Rate Calculation Methods

Terramodel has three methods for calculating flows, each summarized below:

Q Summation

This method lets you use the **Node** command to specify Q values at each node, then computes the flows through each pipe by adding together the Q values of all nodes that are upstream of that pipe. **Terramodel** stores each Q value as an attribute of the node, and updates the values as necessary.

Rational Method ($Q=CIA$)

For each node, you enter an area (A), time of concentration (T_c) for the area that drains to the node, and the ground cover factor (C). **Terramodel** interpolates the rainfall intensity (I) value using the T_c , a storm frequency (that you designate in the **PipeDef** command) and information contained in an external, regional rainfall database file.

Time of Concentration (T_c)

The Rational method adjusts the time of concentration based on the longest travel time to the point of analysis. For this option, **Terramodel** begins the determination of Q at the most upstream nodes, using the maximum T_c for the most upstream node on a branch. For a single pipe entering a node, **Terramodel** computes the maximum T_c as T_c of the upstream node plus the travel time in the pipe. When multiple pipes enter a node, **Terramodel** calculates the longest travel time to the node (also considering any drainage areas that enter the node directly) and uses that value to interpolate the rainfall intensity. **Terramodel** stores each calculated T_c and rainfall intensity as an attribute of their respective node.

As long as your flow rate calculation method is $Q=CIA$, **Terramodel** will update the intensity attribute each time you use the **PipeSolve** command to analyze the system. You can review this value at each node using the **Node** command.

Area

Terramodel determines the total contributing area for each node by adding together the areas that drain to all upstream nodes and saving the total as an attribute of the node.

Ground Cover Factor

For each node, **Terramodel** calculates a weighted C factor and stores it as an attribute of the node. To calculate a weighted C factor, **Terramodel** multiplies each node area by the node C factor, adds the product to the products for all upstream nodes, and divides the sum by the total area.

Calculating the Flow

For each node, **Terramodel** multiplies the weighted C factor, rainfall intensity and total area to determine the Q for the downstream pipe, then stores this value as an attribute of the pipe.

In order to determine the T_c maximum value, you must determine the velocity in each pipe so that you can compute the travel time in each pipe. Although the flow calculations must begin at the most upstream ends of the system and proceed downstream, **Terramodel** must begin the water surface profile and pipe velocity calculations downstream and proceed upstream. You have to calculate the Q values in order to analyze the system, but you can't determine Q values until the velocities are known. There is no single step direct solution to this problem.

Most text books recommend that you use the velocity produced by Manning's equation to estimate the travel time, but this method produces totally invalid results for pipes with horizontal and adverse slopes. You can also assume the pipe is full and estimate the velocity based on full flow, but these velocities may be 4 or 5 times slower than actual results if the pipe is not full. The problem is that for all flow conditions except enclosed flow, the velocity is constantly changing and can only be accurately determined by calculating the true water surface profile.

In order to provide you with a technically correct solution, we've made the following provisions. For each pipe, **Terramodel** stores an average velocity as its attribute. You have complete control of this attribute, which **Terramodel** uses to determine the travel time in the pipe and the T_c maximum for each node. If the value is zero, the travel time is implied to be zero. Here's how you can use this average velocity to determine flows:

1. Use the **Pipev** command, Zero option to change the average velocity for all pipes to zero.
2. Use the **Pipesolv** command to analyze the system. **Terramodel** will calculate the Q values using the travel time in the pipe as zero, and compute the water surface profile. Each time **Pipesolv** is used the average velocities are computed and stored.
3. Use the **Pipesolv** command again for a final analysis. **Terramodel** will use the average velocity to determine the travel time in each pipe and new Q values. You can continue to establish a revised velocity and resolve the system, but our tests have shown that additional trials change the results by less than 1%.

We recommend using this method for a final analysis and not for sizing the pipes and setting the pipe grades, since continuous adjustments to the travel time make only small differences in the calculated Q values.



For this flow rate calculation method, many situations will produce decreasing flow rates as the water travels downstream. This can occur when, for example, there is no flow into a node of a single-branch system. In this situation, the T_c maximum increases and so the I value decreases, causing the flow in the downstream pipe to be less. You should evaluate each design on its own merits and choose an appropriate flow calculation method.

None

This method uses the Q values entered with the **Pipeq** command in calculations. It does not automatically update the pipe Q values.

The None method can be used in conjunction with the other two methods. If you select one of the other flow rate calculation methods to solve the branch, then change to the NONE method, **Terramodel** uses the values calculated by the first method as the default values in the **Pipeq** command. The None option gives you complete control and responsibility for the Q values. When you use this method, you should always verify the Q value used by the **Pipelbr** command.

How to Assign Pipe Sizes

Pipe sizes are defined in an external ASCII text file called `TMPIPE.SIZ` (`TMPIPE_M.SIZ` for metric display units). This file contains various standard pipe sizes and, because it is in ASCII format, you can edit it to add, delete or modify typical sizes. A single line defines the shape, name and dimensions of the pipe for each pipe size.

Pipe Shape

The pipe shape is indicated in the file by the letter C (Circle), R (Rectangular), A (Arch), or T (Trapezoidal). These shapes are the only ones **Terramodel** supports at this writing.

Pipe Name

Each pipe name must be unique to a particular pipe shape. **Terramodel** uses the name of the pipe (maximum 14 characters) on printouts and in the labels of the pipe profiles. Each shape character and pipe name must be followed by a forward slash (/). Names can have no trailing spaces before the slash or after the last character.

Pipe Geometry

The pipe geometry is defined differently depending on the pipe shape. The pipe dimensions for each type of pipe are contained in Table 11-1.

Table 11-1 Pipe Shape Required Dimensions

Pipe shape	Dimensions required (in feet)
Circular	Diameter
Rectangular	Width, Height
Arch	Span, Rise, B, R _c , R _t , R _b ¹
Trapezoidal	Width, Height, Slope ₁ , Slope ₂ , Depth, Slope ₃ , Slope ₄ ²

The dimensions of the shape must be separated with spaces.

Typical open channel trapezoid shapes are shown in Figure 11-1 on page 399. The width parameter for trapezoids is measured from the lowest point at each sideslope.

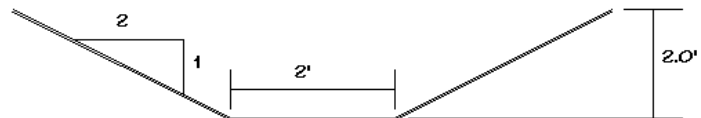
Terramodel uses the height parameter only when drawing the top of the channel with the **Pipedraw** command. Each channel has a single Manning's N value based on the pipe shape/material in the TMPIPE.TBL file.

¹ These dimensions can be found in most publications that list the commercially available sizes.

² Slope parameters are defined in terms of rise over run (i.e., a 4:1 slope would be entered as 0.250). For the purposes of calculations, sideslopes are considered to extend indefinitely.

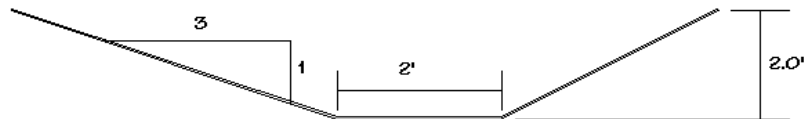
OPEN CHANNEL TRAPEZOID SHAPES

Symmetrical Trapezoid



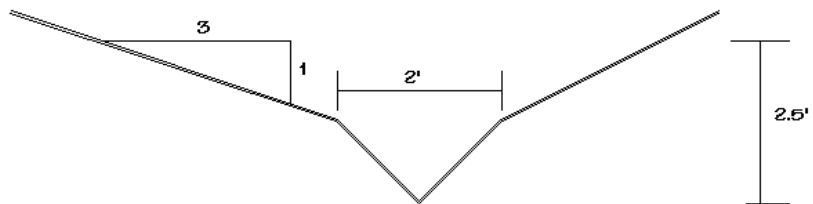
Width 2.0, Height 2.0, Left slope 0.5, Left bottom slope 0.0
Side depth 0.0, Right bottom slope 0.0, Right slope 0.5

Unsymmetrical Trapezoid



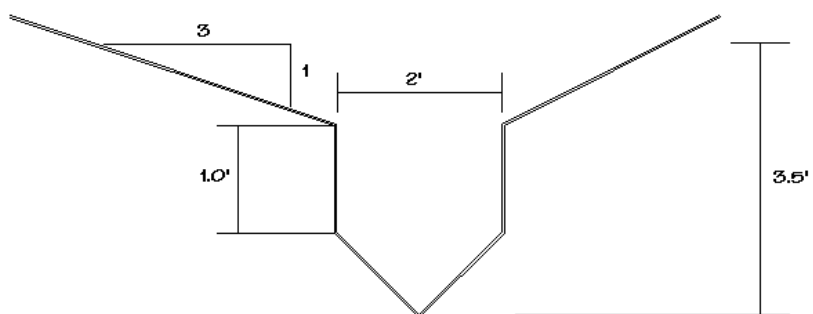
Width 2.0, Height 2.0, Left slope 0.33, Left bottom slope 0.0
Side depth 0.0, Right bottom slope 0.0, Right slope 0.5

Unsymmetrical Trapezoid with Vee Bottom



Width 2.0, Height 2.5, Left slope 0.33, Left bottom slope 1.0
Side depth 0.0, Right bottom slope 1.0, Right slope 0.5

Unsymmetrical Trapezoid with 1' Depth and Vee Bottom



Width 2.0, Height 3.5, Left slope 0.33, Left bottom slope 1.0
Side depth 1.0, Right bottom slope 1.0, Right slope 0.5

All parameters must be positive values

Figure 11-1 Open Channel Trapezoidal Shapes

Designing a Sewer System

Determine the location of the pipe in the plan view.

Create a set in the plan view that properly locates the sewer system. Each point object in the set should represent a node, and the elevation of the point will be used to set the elevation of the top of the node when **Terramodel** creates the node attributes. (If you want to interpolate the elevations of the points from a DTM surface, you can use the **DTMpts** command). You must create the set starting at the most downstream node location.

Create the existing and proposed groundline profiles.

In order to determine the elevations for a pipe system, you have to know the groundline above the pipe. If you have DTMs stored in **Terramodel** for the existing and proposed conditions, you can create the groundline profiles using the **Profile** command. If not, you can store them as polylines in the profile view, entering the stations and elevations.



If the points that form the branch set have elevations and the branch is not a simple straight line, you can profile the layer that contains the branch points, creating a groundline profile from top of node to top of node. This profile can help you to locate the nodes in the profile view.

Establish the defaults for the design.

Use the **Pipedef** command to establish the default hydraulic parameters that **Terramodel** will apply to design the pipes and nodes.

Design the branch.

Use the **PipeDes** command to let **Terramodel** design the branch, applying the default node and pipe parameters. **Terramodel** can design holding either the grade or cover you specified in **PipeDef** command.

Apply the flow parameters to the nodes.

Depending on the flow calculation method you want to use (discussed earlier on page 394), enter the direct flow values or rational method parameters for each node using the **Node** command.

Also enter the tailwater elevation and the velocity for the most downstream node in the system. Since these values are critical to properly modeling the system, you should locate or develop the rating curve for the channel into which the node discharges.

Solve the branch.

Once **Terramodel** has designed a pipe and node system, you can analyze it with the **PipeSolv** command. This analysis process computes flows at the nodes and in the pipes, and calculates the energy and hydraulic grade lines.

If you have one or more branches discharging into a main branch, design and solve the main branch first, then repeat the process with the tributary branches. As long as you use a common point object as the connecting node, **Terramodel** will automatically use the hydraulic grade line for the main branch as the tailwater elevation for the tributary branch(es).

Draw the initial design.

Use the **Pipedraw** command to let **Terramodel** draw a profile in the profile view of the pipe design. This profile will locate the nodes and graphically show you if the pipe sizes were adequate.

Terramodel draws nodes using a 0.5-foot thickness. To graphically represent the pipe-node connections, **Terramodel** clips the pipes to the outside of the nodes.

Draw crossing pipes.

Use the **PipeXing** command to draw in the profile view any pipes that cross the main branch. Drawing crossing pipes helps to identify any potential conflicts.

Edit the nodes and pipes in the system.

Since **Terramodel** designs using only one pipe size (specified in the **PipeDef** command), chances are you'll need to modify the sizes and elevations of the pipes to make them more appropriate for the amount of flow going through them. You may also need adjust the pipes to avoid conflicts with crossing pipes. Use the **Node** and **Pipe** commands to change the nodes and pipes as needed. The **PipeBycr** command lets you hold pipe sizes while changing the pipe elevations, after which you can change the node elevations.

Re-analyze the system.

Use the **PipeSolv** and **Pipedraw** commands to re-analyze and re-draw the system after you make your changes. Each time you view the adjusted pipe profiles, you can see how your changes affect the pipes, nodes and grade lines for the entire system. Finding the right combination will inevitably involve a series of trials of editing and solving, but you'll see how quickly the process goes and how easy it is to make changes.

When you edit the pipes and nodes, work starting from the downstream end and working upstream, since the energy and hydraulic grade lines are calculated in that manner. Whatever changes you make downstream will affect the grade lines upstream.

Print your reports.

Use the **PipeSum**, **Pipelbr**, and **PipeWsp** commands to generate your hydraulic reports. Each of these reports are displayed in the report editor, where you can edit or print them.

Prepare the plan and profile for plotting.

When you've completed your design you can display the plan and profile portions of the system together using dynaviews. You can create a dynaview for the plan portion of the system, and another for the profile portion, and align the two one above the other in the sheet view. Follow the guidelines for creating dynaviews in chapter 6, **Understanding Terramodel Objects**, "Understanding Dynaviews".

Analyzing an Existing Sewer System

Determine the location of the pipe in the plan view.

Create a set in the plan view that properly locates the existing sewer system. Each point object in the set should represent a node, and the elevation of the point will be used to set the elevation of the top of the node when **Terramodel** creates the node attributes. (If you want to interpolate the elevations of the points from a DTM surface, you can use the **DTMPTS** command). You must create the set starting at the most downstream node location.

Create the existing and proposed groundline profiles.

In order to determine the elevations for a pipe system, you have to know the groundline above the pipe. If you have DTMs stored in **Terramodel** for the ground above the pipes, you can create the groundline profiles using the **Profile** command. If not, you can store them as polylines in the profile view, entering the stations and elevations.



If the points that form the branch set have elevations and the branch is not a simple straight line, you can profile the layer that contains the branch points, creating a groundline profile from top of node to top of node. This profile can help you to locate the nodes in the profile view.

Configure the nodes and pipes in the system

Use the **Node** and **Pipe** commands to enter the node and pipe sizes for each structure and pipe segment, as well as the contributing flow to each node. See page 394 for information on flow rate calculation methods.

Solve the pipe system

Use the **PipeSolv** command to analyze the entire system (all branches simultaneously). This analysis process computes flows at the nodes and in the pipes, and calculates the energy and hydraulic grade lines. As long as you use a common point object as the connecting node, **Terramodel** will automatically use the hydraulic grade line for the main branch as the tailwater elevation for the tributary branch(es).

Draw the branches

Use the **Pipedraw** command to let **Terramodel** draw a profile in the profile view of your pipe branches. Draw a branch at a time so you can locate them.

Terramodel draws nodes using a 0.5-foot thickness. To graphically represent the pipe-node connections, **Terramodel** clips the pipes to the outside of the nodes.

Print your Reports

Use the **PipeSum**, **PipeLbr**, and **PipeWsp** commands to generate your hydraulic reports. Each of these reports are displayed in the report editor, where you can edit or print them.

How to Use the Hydro Commands

This section describes the general concepts of how you can use the *Hydrology* module to complete a stormwater detention study. Most of the commands for this module are in the **HYDRO** menu and are secured to this module.

Understanding Hydrology Concepts and Terms

Simply put, the HYDROLOGY module computes basin flows and hydrographs, routes ponds, and creates the associated graphs. Once this information is calculated, **Terramodel** stores it as an attribute of a selected **Terramodel** object - i.e., the basin parameters and generated hydrographs are associated with a point object. Storing data as attributes of objects allows the data to remain with the project file, which means that if you open the file a week later, the basin parameters don't have to be re-calculated. You can also write new TML commands to manipulate the data. And you can modify portions of the design and not have to recalculate the entire study (an example of this would be to update a pond and not have to regenerate its inflow hydrographs).

The HYDROLOGY module also provides flexibility in how you generate reports. During the design phase, you can produce summary reports to help you assess your design. After completing the design, you can prepare detailed reports and listings for submittal to a reviewing agency. Additionally, graphs can be directed to your Windows printer or plotter devices or even exported in a DXF format for inclusion in a word processing package.

The commands that relate directly to the HYDROLOGY module can be found on the **HYDRO** menu, and are discussed in detail in the **HYDRO Menu** section of On-Line Help.

The term *basin* in **Terramodel** refers to a tract of land that drains into a waterway at a point of study. The primary purpose of a detention study is to insure that the peak *post*-development flow rate for a basin does not exceed the peak *pre*-development flow rate. As part of the analysis, the HYDROLOGY module considers the undeveloped portions, or *offsite* areas³, of a drainage basin. Also considered are areas of the site that you are

³ Although the runoff from this area will not be increased by the design, it must be considered because the offsite area also drains into the point of study.

unable to direct into the pond, called pond *bypass* areas⁴.

A successful design is achieved when the peak flow rate from the pond added to the peak flow rate of the bypass area is less than or equal to the pre-developed peak flow rate. The HYDROLOGY module has been designed to enable you to account for any one or all of these areas during the calculations.

Hydrograph Calculation Methods

The following calculation methods are used by **Terramodel** to compute hydrographs for a given basin.

SCS Method

The SCS calculation method used by **Terramodel** employs the same methodology to generate a runoff hydrograph as the Soil Conservation Service's TR-20 method. This portion of the Hydrology module has been tested extensively and the results compared against the PC version of TR-20. For curve numbers over 70 and with typical ranges of precipitation, the peak flow rates between the two programs deviate, on the average, by no more than one percent. Extreme ranges of curve numbers and precipitation values agree to within three percent. The primary reason for the deviation between peak flow rates is that TR-20 reports an additional interpolated peak between data points, while **Terramodel** only reports values at the user-specified intervals, typically every 5 minutes.

When comparing **Terramodel**'s SCS option to studies done using the TR-55 method, be aware of the differences between TR-55 and TR-20. The original tabular method of TR-55 was developed by making several computer runs with the TR-20 program. In each case, a runoff curve number (CN) of 75 was used with rainfall volumes such that three inches of runoff volume was generated. Any deviation from the data points used to create the charts and graphs for TR-55 will produce results that vary from TR-20. Although the 1986 update to TR-55 added charts to cover a larger range of values, users are still forced to base their calculations on a linear interpolation of charts for calculations that are non-linear. The 1986 TR-55 update lists several conditions for which TR-55 should not be used, and recommends using TR-20 for more accurate results. The TR-55 method was created to emulate TR-20 using charts and graphs, and any difference between the two reflects the inaccuracy of TR-55.

⁴ This portion of the site may have changes in runoff as a result of improvements, and must be accounted for in the design.

Mistakes in the selection of curve numbers is one cause of inaccurate results using the SCS method. Unlike the rational method, some combinations of precipitation and curve numbers will actually generate no runoff. Small amounts of runoff are spread over the duration period (6/24/72/120 hours) and may produce peaks that are unexpectedly low. Unlike the rational method, where C factors have a linear effect on the peak flow rates, curve numbers do not produce linear results.

The following table demonstrates how small variations in curve numbers affect the amount of direct runoff (DRO). For additional information regarding the SCS method, refer to the references listed at the end of this chapter.

Table 11-2 Effect of Variations in CN on Direct Runoff

P(in)	CN 25 DRO(in)	CN35 DRO(in)	CN50 DRO(in)	CN 75 DRO(in)	CN 95 DRO(in)
1	0.00	0.00	0.00	0.00	0.56
2	0.00	0.00	0.00	0.38	1.48
4	0.00	0.00	0.33	1.67	3.43
6	0.00	0.25	1.14	3.28	5.41
8	0.13	0.80	2.25	5.04	7.40
10	0.47	1.59	3.56	6.88	9.39

Universal Rational Method

“Universal Rational” is a term coined by Trimble, and was first made available in the HYDRO/PLUS program in 1983. This method is based on the Baumgardner/Morris FHWA method published in 1982, and provides an approximate procedure for developing a runoff hydrograph using the rational equation, $Q=CIA$. When using this option, you have the choice of placing the hydrograph peak at either $1xT_C$ or $3xT_C$. By placing the peak at $3x T_C$ (the recommended option) you can achieve a more realistic (conservative) design. Increasing the amount of flow that occurs before the peak enables the pond to partially fill before the peak flow rate occurs, more closely emulating the rainfall distribution used by the TR-20 method.

The Baumgardner/Morris FHWA method is also described in the State of Florida Department of Transportation drainage manual. We can provide a copy of the original FHWA paper upon request.

We recommend you limit this method to basins of less than 500 acres and times of concentration of less than 30 minutes. The valid minimum and maximum times of concentration are 5 minutes and 144 minutes, respectively.

DeKalb Rational Method

The DeKalb Rational method was developed for use within the metropolitan Atlanta area. The methodology is based on the rational formula and unit hydrograph theory, as detailed in the DeKalb County Drainage Procedures Manual.

In developing this methodology, flood hydrographs were created using the rainfall database curves (IDF curves) by computing incremental rainfall intensities and arranging them to form a storm pattern.

Two typical dimensionless hydrographs were developed for use in the Atlanta area by DeKalb county. **Terramodel** selects the appropriate hydrograph based on the time of concentration. Each ordinate is then simply a percentage of the peak, where the peak is based on $Q=CIA$.

The peak flow rate always occurs at $5xT_c$. the base of the hydrograph extends to $10xT_c$. We recommend you limit this method to basins of less than 500 acres and times of concentration of less than 30 minutes. The valid minimum and maximum times of concentration are 5 minutes and 144 minutes, respectively.

The table below shows the dimensionless hydrographs that are used with the DeKalb Rational method:

Table 11-3 DeKalb Rational Dimensionless Hydrographs

t/t_c	t_c < 20 minutes	t_c ≥ 20 minutes
1	0.16	0.04
2	0.19	0.08
3	0.27	0.16
4	0.45	0.32
5	1.00	1.00
6	0.34	0.30
7	0.27	0.11

8	0.19	0.05
9	0.12	0.03
10	0.00	0.00

Designing a Simple Basin

Construct a site plan.

The basic requirements for a site plan include:

- Topography or contour lines for the area being studied, including the pre-developed and developed conditions
- The types of ground cover for the pre-developed and developed conditions
- The proposed location for the detention pond
- Any associated storm drainage facilities

Your site plan can be created using **Terramodel** (the preferred method), imported from another program, or even drafted by hand.

Establish the default hydrology settings.

The **Hdef** command establishes the rainfall database and calculation method to use, storm return periods and other parameters required to calculate hydrographs for a basin.

Each geographic region requires different rainfall intensity, duration, and frequency (IDF) information, which you enter into an ASCII file that is referenced by the HYDROLOGY module. The rainfall data for the Atlanta region has been supplied in the ATLANTA.RNF, and is described in the **HYDRO Menu** section of **Terramodel On-Line Help**.

Determine the drainage areas.

Use the **DtmDrain** command to outline drainage areas with set lines and calculate the area values. Once you have verified that each area is accurately defined, use the area values for defining the basin.

You can also use other **Terramodel** commands or manual methods to determine the drainage areas.

Locate the basin/pond.

Place a point object at the pond location or the lowest elevation on the basin. The location of the point is not critical, but is used to graphically indicate the point of study. All of the attributes for the basin and pond will be attached to this point. If your site plan is not part of the current project file, the point can be placed at any location.

It's helpful to place the point on a unique layer to designate its usage, and to label the point with a symbol to identify it as a basin.

Determine the C/CN factors.

Classify the types of groundcover for the pre- and post-development conditions, then determine an average “C” factor or curve number for the offsite, pre-developed, developed and bypass areas. You can use your own methods to calculate a weighted coefficient or, using the **Hccn** command, select the point established earlier as the basin and enter the proportions of three types of ground cover. Given the coefficient or curve number and the area for the impervious, landscaped, and natural areas, **Hccn** calculates the weighted value, assigns the weighted value as an attribute of the basin, and creates a report. Since **Hccn** is a TML command, you can customize the description of the types of cover.

Determine the time of concentration (T_c).

There are many methods used to determine the time of concentration for a basin. The HYDROLOGY module doesn’t provide any specialized commands to determine T_c . If the site plan is part of the current project file, you can use the **Inverse** command to determine the linear or slope distance that a drop of water will have to travel along the surface. You can also use the **Spot** command to determine the elevation at various points on the DTM. A time of concentration can be calculated by knowing the length and average velocity.

If you have a specific, preferred method, you can write a TML command to calculate the value and store this as an attribute of the basin. **Terramodel** requires a T_c value for all pre-developed, developed and by-pass areas.

Define the basin parameters.

The **Basin** command defines the parameters for the basin. Each basin contains attributes to define the pre-developed, developed and by-pass conditions. For each of these conditions you will specify:

- The area.
- The average basin slope.
- The hydraulic length.
- The time of concentration.
- The factor or curve number (depending on the method used for analysis).

Use the **Hareab** command to calculate the area of a line (set line, polyline or breakline) and assign the area to the pre-developed, developed or bypass areas of a basin. The average basin slope and hydraulic length are used for reporting purposes only and will not affect the calculations. The **Hccn** command can define the C/CN and area parameters for you, or you can enter the calculated values manually using **Basin**.

Generate, list, and draw the hydrographs.

Use the **Hgen** command to generate the hydrographs and produce a summary in the report editor showing the frequency, pre-developed peak flow, bypass flow rate, developed flow rate and the allowable peak flow rate. The entire hydrographs and their summary values are also stored as attributes of the basin.

The **Hblist** command lists the summary information for the basin and the summaries of the computed hydrographs to the report editor. You can also use it to list the detailed hydrographs for the basin.

The **Hdraw** command draws the basin hydrographs, creating text and polylines in the sheet view mode. You can modify these plots with polylines and text if you like.

Design the Pond.

If you created your site plan in a **Terramodel** project file, you can use the **SITE DESIGN** module to design the grading plan for the pond. A good starting point is to make the pond hold 30% to 40% of the volume of the developed hydrograph. This volume will vary greatly depending on the percentage of change in runoff from the pre-developed to developed condition and the efficiency of the outlet structure.

You'll need the following information to design your pond:

- The invert elevation of the outlet structure.
- The elevation and area of the pond at the invert, for a significant number of intervals to describe the geometry of the pond.

Define the pond geometry.

The **Pondvol** command calculates and/or stores pond volumes as attributes of the basin/pond point. You can enter elevations and areas (they don't have to be entered in sequential order-they are automatically sorted) and let the program use the average end area method to compute volumes, or you can enter the elevations and incremental volumes directly. You can also use the **Inpond** command to compute pond volumes from closed contours in the project file and store the volumes as attributes of the pond in the **Pondvol** command.

Define the pond outlet devices.

Each pond can have up to ten outlet devices, including a user-defined device. Typical devices include V notch weirs and orifices. Pay particular attention to the selection of coefficients and the equations used, and also consider that these equations do not consider any tailwater effects. Tailwater effects can best be handled by entering the device elevations and discharges manually.

The **Pondout** command stores the pond outlet devices. Its **Design** option accepts a desired flowrate at a desired depth, then displays the appropriate size outlet device to accommodate it. You can then round the value to a commonly fabricated size, if necessary. The **Design** option is not available for all of the device types.

List the pond parameters.

The **Pondlist** lists the stage-storage-discharge data for the pond and will optionally generate a graph in the sheet view of the stage versus discharge and storage values.

Route the hydrographs through the pond.

After the basin and pond parameters have been defined, use the **Hrout** command to route the basin through the pond. The **Pondsum** command provides a summary of the pond routing.

If the maximum pond elevation is exceeded, you will be warned but the computations will continue. You should either revise the pond geometry to provide more storage or provide a more efficient outlet device.

Plot the inflow outflow hydrographs.

Use the **Hdraw** command to draw the inflow and outflow hydrographs in the sheet view.

Assemble a complete report.

Typically, different portions of the design are accomplished by an iterative process. Once you've completed a satisfactory design, you can use the commands listed below to prepare a final report.

Table 11-4 Hydrology Module Commands

Use	To
HCOVER	Generate a cover sheet for a hydrology report.
HDEF	Establish defaults for hydrograph calculation.
DTMDRAIN	Outline drainage areas.
HCCN	Compute weighted coefficients or curve numbers.
BASIN	Define the design parameters for a basin
HGEN	Generate the pre-developed and developed hydrographs.
HDRAW	Plot the pre-developed and developed hydrographs
HLIST	List the pre-developed and developed hydrographs.
PONDVOL	Store the pond elevation, area/volume pairs.
PONDOUT	Enter the pond outlet devices.
PONDOUT; List option	List the pond outlet devices.
PONDLIST	Plot the pond stage/storage/discharge curve.
HROUT	Route the basin hydrograph through the pond.
HDRAW; Outflow option	Plot the inflow and outflow hydrographs.
PONDSUM	Show a summary of how the pond operated for each storm event.

Designing Multiple Basins

Multiple basin design is used when a single large basin requires two or more detention ponds to control the stormwater, and one pond drains into another. Two or more ponds that drain offsite independently should be considered two independent basins.

In a simple multiple pond design, the upper portion of a basin drains into a pond. The pond drains into a reach which, along with the lower portion of the basin, drains into a second pond. To achieve a proper design, the outflow from the lower pond must not exceed the pre-developed flow for the entire basin.

The basic procedure for multiple basins is as follows:

1. Place point objects at the pond locations.
2. Connect the two pond points together with a set line. The downstream pond point should be selected as the first point in the set to establish the direction of flow.
3. Size each pond as discussed as you would for a single basin.
4. Using the **Reach** command, assign Muskingum parameters to the set. For most basins it is recommended that the k and x parameters be set to zero. When set to zero the hydrograph will be translated through the reach with no peak attenuation. This is a more conservative approach.
5. Generate the basin hydrographs.
6. Route the upstream pond, then the downstream pond. The **Hrout** command adds the basin hydrographs to the hydrograph routed through the reach and routes the combined hydrographs.
7. Use the **Hrlist** command to list the effects of the reach routing. The pond inflow hydrograph will be the combination of the two inflow hydrographs.

You can include additional basins, working toward the downstream direction and generating the hydrographs for each basin before routing the basin hydrographs through the pond.

Engineers have raised theoretical questions about the validity of adding hydrographs generated by rational methods when using different time of concentration values. Rational methods are typically used for small basins and should be limited to T_c values of less than 30 minutes. Because rational methods use the time of concentration to determine the peak, duration and shape of the hydrograph, mixing hydrographs with different T_c values may not give valid results. If given the choice, use SCS methods where each hydrograph is generated using the same duration of rainfall. Hydrograph addition using SCS methods is an accepted practice.

Customizing the Hydrology Module Using TML

Much of the HYDROLOGY module was written using the **Terramodel** Macro Language (TML). The commands used in the HYDROLOGY module are actually external ASCII files that you can edit. Using TML allows you to make simple changes (such as changing the number of decimal places in a report) as well as more complex changes (actually adding another hydrograph generation method!). **Terramodel** with TML makes your possibilities for stormwater detention virtually unlimited.

The HYDROLOGY module takes advantage of the TMLs and object attributes to create a comprehensive and extensive system. For the average user, TML and the attributes simply work as a part of the system and require no specific knowledge. The ultimate goal of using TML and attributes is to make enhancements easy and to allow advanced users to customize the design.

If you want to customize or add to the HYDROLOGY module, first study the TML commands used by the HYDROLOGY module and refer to the TML help system as needed. **Terramodel** stores all data required for the HYDROLOGY module calculations and the calculated hydrographs as attributes using the attribute definition file HYDRO.ADF. These attributes are complex and should only be revised by someone with programming knowledge.

HYDROLOGY Module References

The following references were used as sources of information for **Terramodel**'s hydrology and hydraulics calculations:

- 1 Brater and King, Handbook of Hydraulics, Sixth edition, McGraw Hill, 1982
- 2 Chow, Ven Te, Open Channel Hydraulics, McGraw Hill, 1963
- 3 Dekalb County, Drainage Procedures Manual, 1976

Available from:

DeKalb County Roads and Drainage Department
1 Callaway Square
Decatur, GA 30030

- 4 McCuen, Richard H., A Guide to Hydrologic Analysis Using SCS Methods, Prentice Hall, 1982
- 5 Soil Conservation Service, Urban Hydrology for Small Watersheds (TR-55 2nd Edition), National Technical Information Service, 1986.
- 6 Soil Conservation Service, National Engineering Handbook, Section 4 Hydrology, NTIS, 1969.
- 7 State of Florida Department of Transportation, Drainage Manual, 1987.

Available from:

Florida Department of Transportation
Roadway Design Office
Mail Station 32
605 Suwannee Street
Tallahassee, FL 32301

- 8 United States Department of the Interior, Design of Small Dams (2nd Edition), NTIS, 1977
- 9 Vennard and Street, Elementary Fluid Mechanics (Fifth Edition) John Wiley & Sons, Inc. 1975

Chapter Twelve

Automated Sheet Assembly and Production

Getting Started

Using the Plan Set Commands

The Terramodel *Automated Sheet Assembly and Production* (ASAP) module gives you the power to assemble and manage an entire set of construction plans or maps. The ASAP module addresses two important areas of drafting and project management productivity:

- Creation of individual drawings.
- Assembly of drawings into a coherent and organized set of plans.

The ASAP module automates the process of creating finished plan sheets. It assembles the various drawing elements, such as the border and title block, dynaviews (of plan, profile, or cross-section drawings), north arrows, titles, labels, sheet numbers, etc.

Although **Terramodel** creates the drawings for you, you have complete control over their content and format.

Sheets are created in **Terramodel**'s sheet view mode. They are arranged in the order in which they will appear in the final plan set. You can create new sheets and place them at any desired position, delete existing sheets, or change the order of sheets within the plan set. As you do, **Terramodel** automatically changes the sheet numbers to reflect the new plan set content and order. Other textual references to sheet numbers (such as a matchline label which calls out the sheet number of the adjacent sheet) are updated as well. This applies to textual references which the software creates, and to notes that you might place on one sheet referring to another sheet.

If you like, **Terramodel** will create an index of drawings, which you can place on any sheet you wish. This index is a table containing the sheet number, drawing title, and other pertinent information for every sheet in the plan set. Every time you add, delete, or move a sheet, the drawing index is automatically updated to reflect the current content and order of the plan set. Also, if you change the title of a drawing, that change is instantly reflected in the index of drawings.

Thanks to **Terramodel**'s *smart text* feature, changes to project information (such as the name of a roadway alignment) can automatically be reflected in the text objects, which label that roadway's name. This can occur within the drawing index, and everywhere such text exists throughout the entire plan set. **Terramodel**'s Project Data Editor, discussed in the following section, extends this concept to include project data not associated with graphical objects. Information such as the project name and location; job number; client name, address, and phone number; and staffing information such as the project manager, engineer, or surveyor of record, etc., are stored within the project file. You can link text placed within the drawings to this data. You can easily make any desired changes to the project information within the Project Data Editor's dialog boxes. As a change is made in that editor, all linked text throughout the entire plan set is updated to reflect that change.

Terramodel's features not only greatly enhance your productivity in creating drawings, but also help you avoid numerous sources of potential errors as those inevitable changes creep into your project (and attempt to consume your drafting budget).

Although the ASAP module utilizes some of the more advanced features of **Terramodel**, and these are explained in detail for those interested, an advanced level of expertise is not required to effectively use these commands. This chapter does however assume that you have mastered the basic concepts of using **Terramodel**.

Using the ASAP commands

The ASAP module commands are accessed via the **DRAFT** menu. There are two major types of commands that are used in the assembly of a set of plans, those that manipulate the set of plans and those that manipulate the project information that is used in the set of plans. The two menu selections are *ASAP-Plan set* and *ASAP-Project data*.

Selecting **ASAP-Plan Set** from the **DRAFT** menu opens a submenu containing the following seven selections:



Figure 12–1 The Plan Set Submenu

Each of the commands work together to create a set of plans. Because of the close association of these commands you can also enter **planset** as a command so that all the above commands can be selected from the planset command bar as shown below.



Figure 12–2 The Plan Set Command Bar

Working with Plan Sets

Plan Set Concepts and Terminology

The Terramodel ASAP module was developed to simplify the process of creating finished drawings. It automates much of the work required in this process. It is both easy to use, and easy to learn to use. There are just a few simple concepts and terms that you must be familiar with.

The term “plan set” refers to the set of drawings, which comprise all of the construction drawings, maps and plans associated with your project. The components related to, and required to assemble a plan set are defined and illustrated below.

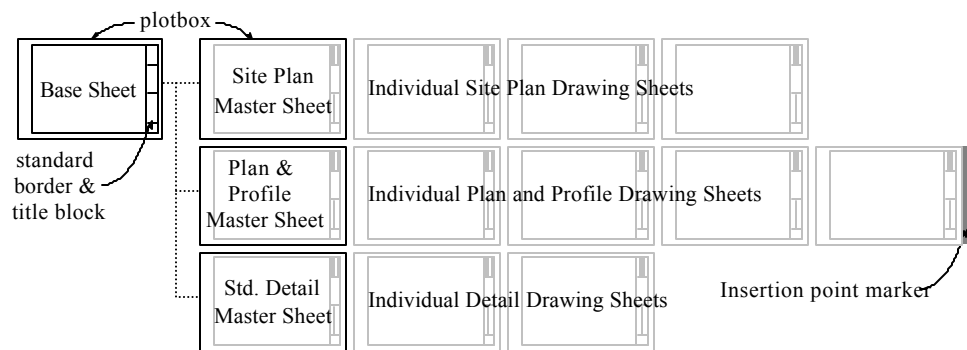


Figure 12-3 Plan Set Terminology

Base Sheet—A base sheet is a drawing within the sheet view mode, which contains the basic elements that are to be shown on every sheet in the plan package, or every associated sheet if there is more than one base sheet. It establishes the size of the associated drawings, and typically will contain a drawing of a standard border and title block as well, establishing this as the look of each associated sheet in the plan set. Figure 12-3 shows a base sheet containing a standard border and title block.

Sheet Queue—A sheet queue is a series of sheets that are associated with each other, generally because they are of a particular type. You can configure the plan set using one or more sheet queues. In the above illustration there are three queues, one containing site plans, one containing plan and profile drawings, and one containing standard detail sheets. Terramodel will arrange all sheets within a queue in a horizontal row in the order

of their sheet number. Each sheet queue is associated with a unique master sheet, as seen in Figure 12-3.

Master Sheet—A master sheet is a special sheet that establishes the common content of each sheet in a sheet queue. A project can contain sheets of varying size, and with borders and titles blocks of varying design, but these features must be consistent within each sheet queue, as defined by its master sheet. Every master sheet references a base sheet, and via dynaview displays all of the graphics displayed within its base sheet. Any additional graphics drawn within the master sheet (in addition to those contained in the base sheet) will be shown via dynaview on every sheet displayed within its queue. Therefore, a general note which you might wish to be included on every plan and profile sheet, need only be placed once on the plan and profile master sheet. Each sheet queue can have an associated drawing ID prefix that is stored as an attribute of the master sheet. Drawing ID's are defined below.

Drawing Sheet—These are the actual sheets, which comprise the set of plans that illustrate the project's design. The **Planset** command automatically creates these drawing sheets. Each drawing sheet, contains a dynaview of the master sheet, thereby replicating the graphics contained within that master sheet, and those within the associated base sheet as well. Additionally, dynaviews of the appropriate portions of the plan and profile view modes, north arrows, profile grids, titles, labels, sheet numbers, drawing IDs and other sheet specific title block entries are placed within the drawing sheet in the appropriate location.

Sheet Insertion Point Marker—The sheet insertion point marker is a filled block, as shown in Figure 12-3, which is located after a drawing sheet or master sheet to mark the location at which the next created sheet will be placed. It occupies the gap between sheets. This concept is analogous to the text insertion point marker in a word processor, which marks the place where the next text entry will be placed. You can place the sheet insertion point marker wherever you wish within the plan set.

Sheet Number—A sequential integer number is assigned to each drawing sheet in the plan set. Sheets are numbered from left to right within a queue, starting at the top queue and proceeding through each lower queue. As sheets are inserted, deleted or moved within the plan set, the sheet numbers are changed to reflect the revised sequence. In addition, references to sheets such as those associated with match lines, are updated in these cases. The sheet number, and the total number of sheets within the plan set can optionally be included in a designated location on each drawing sheet.

Drawing ID—The drawing ID is an alphanumeric ID consisting of a user defined prefix and a sequential integer number representing the sheet's position (starting at 1) within its individual sheet queue. This supports a numbering convention such as PP-5, where the prefix "PP-" indicates a plan and profile sheet and the number 5 indicates the fifth sheet in that queue; or S-2, where the prefix "S-" indicates a site plan sheet, and the number 2 indicates the second sheet in that queue. Sheet PP-5, the fifth sheet in the plan and profile queue, might also be sheet 23 of 46 in the plan set.

Plan Set Configuration Settings

Regardless of the productivity increase achieved, automation of the drawing creation process is of limited value if you are not satisfied with how the drawings look. Configuration settings allow you to control the way ASAP works. All of these settings are stored within the project file, and can be edited. These settings can be stored in a prototype project file, if you wish. They can also be stored in an external binary file, and loaded as needed. Choose the method that works best for you.

You might find that you need a library of plan set settings files. You could have one, which uses your normal in-house drawing standards, and others that conform to standards imposed by your clients. Storing these in an external file and loading them as needed is convenient and a potential time saver. This method eliminates the need to store the settings in your prototype project file, or to edit them continually.

The **Planset** command's settings allow you to control:

- The layers upon which objects are drawn as they are created.
- The layer lists assigned to the dynaviews, which control their displayed content.
- Configuration of the sheets.
- Plotbox dimensions.
- Location of components such as the plan and profile views, raster images, north arrows, etc.
- Wording, formatting, styling, position and presence of labels and titles.
- Drawing and labeling of the profile grid and matchlines.
- Layout, content and format of the index of drawings.

The configuration settings can be edited, loaded, or saved by selecting the **Plan Set/Settings** option (**psset** command), from the **DRAFT** menu. You can find detailed information about **psset** in the on-line help system.

Graphical Editing of Label Settings

Most configuration settings are edited via dialog boxes. This is true of the label settings as well, but they can also be edited graphically. Some of the drawing labels must be precisely placed and oriented within a title block. Determining and entering x and y offset values in these cases can be difficult. Graphical editing of these settings makes this process easy.

To edit the label settings graphically, specify the sheet queue in which you wish to edit labels by placing the sheet insertion point marker within that queue. Click the Show Settings button. The sheet view zooms to the extents of the master sheet for the sheet queue you have specified, and the master sheet is labeled with a sample of each label, placed and formatted in accordance with the current settings. You can then use any of **Terramodel's** editing features to move or rotate the sample text, change its color or text metrics, or edit the text string itself. After visually editing the labels, click the Update Settings button. **Terramodel** retrieves each of the sample labels, edits the label settings to match, and deletes the samples.

You can edit the label settings by selecting the **Plan Set/Settings** option (**psset** command), from the **DRAFT** menu and clicking the Labels button. For more information on the editing of label settings, see the **psset** command, in the on-line help system.

Supported Drawing Configurations

The **Planset** command currently supports the following drawing configurations (sheet types).

- Single plan sheet
- Double plan sheet
- Single plan and profile sheet
- Cross-section sheet
- Miscellaneous Sheet

To assign the current sheet type, select the **Plan Set/Sheet Type** option (**pssheet** command) from the **DRAFT** menu. When you select a sheet type, it remains the current sheet type until you select another. You can find more information on selecting a sheet type in the on-line help system, the **pssheet** command.

Alignment-Based Plan Sheets

This type of drawing is typical of that used for a simple pipeline project for which a profile view is neither warranted nor desired. It simply presents a plan view of the alignment corridor. In many cases, the geometry of the alignment and the corridor width is such that two strips of plan views can easily be placed on one sheet. You may, therefore, create either single or double plan sheets as your needs may warrant.

Alignment-Based Plan and Profile Sheets

The program will create plan and profile sheets, placing either view as desired by the user. The profile views can include a profile grid and labeling. The grid lines and labels are drawn within the drawing sheet in the sheet viewing mode, so as not to clutter the profile view (a design data view) with objects that are most often irrelevant to the design process.

Alignment-Based Cross-section Sheets

Sheets illustrating cross-sections taken from an alignment can be created. These will include the cross-sections of multiple designated surfaces, and the cross-section grid and labeling.

Miscellaneous Sheets

You can create a miscellaneous sheet at any location within the plan set. Such a sheet will include the contents of the master sheet and the appropriate title block information, sheet number and drawing ID, as desired. You can then draw within these sheets, or place dynaviews as needed to meet your needs for any type of drawing.

Creating an Index of Drawings

You can have **Terramodel** automatically create an index of all the drawings contained within the plan set and place this index on any sheet you wish. As the content of the plan set is changed, the index is automatically updated. The index will contain the sheet number, if sheet numbers are used, and the drawing ID, if they are used. It will also contain the sheet title. In the case of alignment-based plans, the index may also contain the alignment name (in addition to, or in place of the sheet title) and the limits of stationing for each alignment on a sheet. If you change the title of a sheet or the name of an alignment, that change is automatically reflected in the index of drawings.

The **Planset** command's configuration settings let you control the content and format of the index of drawings. You can specify:

- The index title.
- Content headings.
- The text style and color of each component.

- Spacing between columns and lines.
- The information to be included for alignment-based drawings.
- Whether or not a border is to be included and, if so, its characteristics.

Create an index of drawings by selecting the **Plan Set/Indexes** option (**psindex** command) from the **DRAFT** menu. Additional information on the creation of a drawing index can be found in the on-line help system, the **psindex** command.

Automated Sheet Assembly

The sheet assembly component of the ASAP module automatically creates most of the civil engineering and surveying drawing sheets in a typical project. It can include a border, a title block with many of the required entries, sheet numbers, titles, various labels, north arrow(s), and dynaviews of the subject areas within the plan, profile and cross-section views as the sheets are created.

The sheet assembly process can include four operations. The first is the creation of the basic sheet. After creating the sheet you can move or rotate the plan view or move the profile view if needed. **This must be done using the **Planset** command so as to assure that all of the objects attached to or associated with the plan and profile views are moved as a unit.** The fourth operation you can perform in assembling a sheet is to register user created objects to the sheet or to a particular view. If you add objects to a sheet, such as notes, details, etc., you must register these objects as belonging to that sheet. This is so that they are moved with that sheet within the plan set as sheets are inserted, deleted, or reordered, or so that they are deleted if the sheet in which they reside is deleted.

Using Miscellaneous or Blank Sheets

In order to take advantage of the plan set management aspects of this module, you will want every sheet in your project's plan set to be created by the ASAP module. For this reason we have designed the software to be as comprehensive as possible in the types of drawings that it can create.

You may, however, occasionally need to create sheets of a type not supported by **Planset** (e.g., a sheet illustrating a group of standard details, or a sheet that combines the index of drawings with an index of abbreviations and an index of symbols). For these purposes, you can use the Miscellaneous sheet type. When you create a miscellaneous sheet, **Terramodel** generates a sheet, which includes the contents of the associated master sheet. It can might also contain a sheet title, sheet number, drawing ID, and the designed by, drawn by, checked by, and approved by names. This produces a sheet

within the plan set, which takes advantage of all of the plan set management features. You may then add the drawing content that you need, and register that content to the sheet.

In addition to the specific sheet types listed above, **Planset** allows you to create blank sheets. The term “blank sheet”, in this instance, means a sheet that has all of the contents of its master sheet (i.e. border, title block, etc.) and, in addition, can have a sheet number, drawing ID, sheet title, etc. When created, however, a blank sheet has no drawing content.

A blank sheet is different from a miscellaneous sheet in one respect. A miscellaneous sheet is of the miscellaneous sheet type. A blank sheet, however, can be of any designated sheet type, such as a single plan and profile sheet. This has implications in the creation of an index of drawings where the format is different for alignment based sheets, as described above. If you have a plan and profile sheet showing two short alignments, and one of those alignments must be lengthened so that they would both no longer both fit on that sheet, you could create a blank plan and profile sheet, and then move one of the alignments to the blank sheet.

When using miscellaneous or blank sheets, you may still take advantage of all of the plan set management features. The drawing content of the sheet, if any, must be inserted manually, however. In the case of a standard detail sheet, you could insert a block illustrating each desired standard detail from your library.

Also, you can insert a blank sheet as a place-holder for a drawing produced by a sub-consultant, or some other source, where it is generated by other means. This lets you create a place for it in the drawing order, include it in the index of drawings, and insure that all the other sheets are numbered properly.

For more information, see “Creating Blank Sheets”, below.

Working with Sheets

Creating a Base Sheet

At least one base sheet must be located within the sheet view mode. A base sheet is simply a rectangular polyline plotbox, which contains the graphics that you wish to appear on every associated sheet in the plan set. The base sheet will typically contain, at least, a drawing border and a title block within its plotbox.

The plotbox itself is used to establish the size of the sheet. You might draw a base sheet plotbox 24 inches high and 36 inches wide, representing the edge of the paper for the finished sheet. Every master sheet, which is based on that base sheet, and every subsequent drawing sheet within its sheet queue, will also contain a rectangular polyline of this size. These polyline boxes serve both as plotboxes for plotting the sheets, and as the basic identifying element of the sheet, with which all other objects within that sheet are associated. In working with the **Planset** commands you will be able to select a sheet either by picking this outline box or by picking anywhere within it. Subsequent operations will treat all of the objects that are members of that sheet as a single entity. **It is, therefore, imperative that you not alter or delete these sheet outline boxes.**

You can find more information on determining the appropriate size of the base sheet plotbox in the **DRAFT Menu**, “Plan Set Management (**psman** command) —Create Master Sheet Command Bar”, in the on-line help system for the **psman** command.

The border and title block can be drawn within the base sheet’s outline plotbox. You can draw a base sheet, complete with a standard border and title block, within your prototype project files. Otherwise, you could store base sheets containing standard borders and title blocks representing several common sheet sizes within external blocks. These could be inserted and exploded, creating a base sheet as needed. If your plan set contains sheets of different sizes, or sheets with differing title blocks, you can create a different base sheet for each need.

At this time, the **Planset** command supports only one plan set per project file.

Place within the base sheet, all title block text that needs to appear on every drawing sheet. Since each drawing sheet contains a dynaview of its master sheet, and the master sheet contains a dynaview of the base sheet, this information automatically appears on every drawing sheet.

When you place the information on the base sheet, you save time, memory and file space by placing the information only once. In addition, this eases the task of editing the title block, if necessary.

Information you might place within the title block of the base sheet includes:

- Project title and subtitles
- Job number
- Department
- Project location information
- Client's name, address, phone and fax numbers
- Staff names

When you place this information in your title block, link the text to the appropriate project data so that it will automatically be updated should that information change. Refer to the following section on **Terramodel's** Project Data commands for a complete discussion of how to link text to project data.

Planset keeps track both of sheet numbers and of the total number of sheets in the plan set. You may wish to number sheets using the convention "Sheet 12 of 46", showing both the individual and total sheet numbers.

To use the "Sheet x of x" format:

- ⊕ Place a single text object within the base sheet title block that includes the word "Sheet" followed by four spaces and the word "of".
- ⊕ Place the sheet number total as a separate left justified text object within the base sheet, after the word "of".

Planset can assign this text object an attribute indicating that it is the sheet number total. Every time a sheet is created or deleted this text object will be updated and, since it is located on the base sheet, it is shown on every associated sheet.

- ⊕ Place the sheet number as a right justified text object on each individual sheet, within the spaces in front of the word "of".

For more information on assigning the sheet number total attribute to a text object, see **DRAFT Menu**, "Plan Set/Settings (**psset** command)—Sheet Number Totalization", in the on-line help system for the **psset** command.

Creating a Master Sheet

After you have created a base sheet, use the **psman** command, *Create Master Sheet* command bar, to create a master sheet. Pick the base sheet on which you wish to base the master sheet, and pick the location where you wish the master sheet to be placed.



*If you wish, you may specify a drawing ID prefix when the master sheet is created. Drawing IDs and prefixes are explained on page 423. Use the **psman** command, *Edit Drawing ID prefix* command bar to enter and edit the drawing ID prefix. Once a master sheet is created you can easily edit its drawing ID prefix.*

You can also delete a master sheet as long as there are no sheets in its queue.

The master sheet management commands can be accessed by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and then selecting the Master button. You can find more information on creating a master sheet, deleting a master sheet, and on editing the drawing ID prefix assigned to a master sheet in **DRAFT Menu**, “Plan Set Management (**psman** command)—Master Sheet Command Bar”, in the on-line help system for the **psman** command.

Creating Plotboxes

Terramodel's sheet view mode and dynaviews are used to compose finished sheets that contain view ports into the plan, profile, and cross-section view modes. The creation of a dynaview requires that you draw a plotbox in the desired view mode, which defines the area you wish to see in the dynaview. See chapter 5, **Understanding Terramodel CAD**, "Working with Dynaviews", for more information on this process.

To create the required plotboxes for a complex project manually can be a complicated and time consuming process. For example, imagine that you need plan and profile drawings for a subdivision consisting of several curvilinear roadways. Below is an example showing all the roadway centerlines and the desired plan view plotboxes for just one of those roadways.

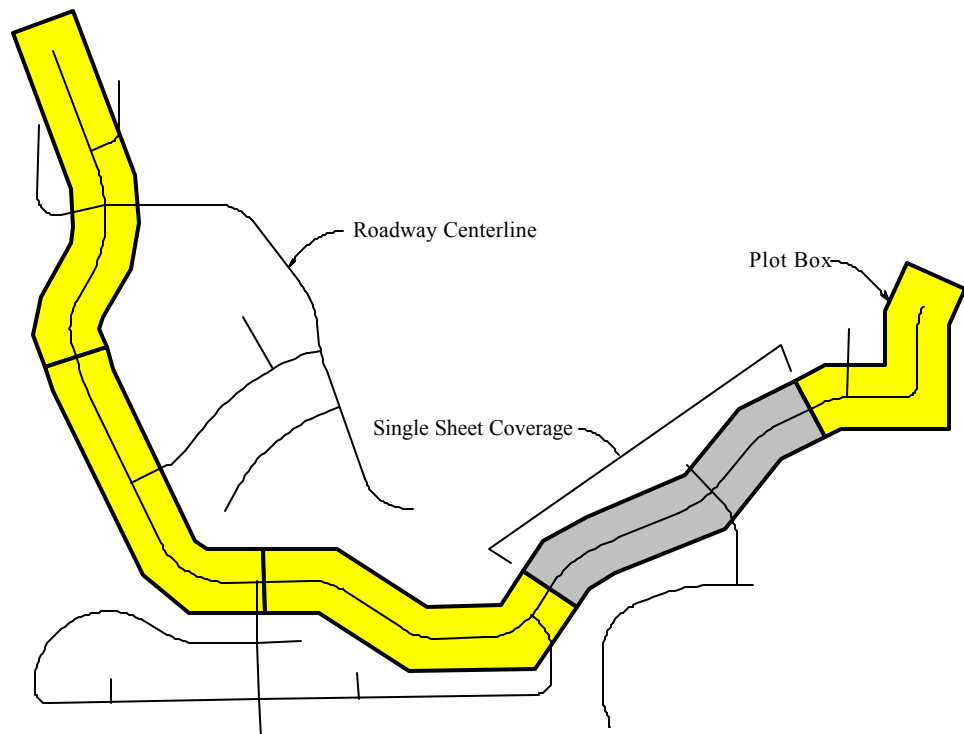


Figure 12-4 Typical Curvilinear Roadway Plotbox Shapes

To create the plotboxes manually, you would have to know how many inches of profile will fit on your sheet, then convert that figure to feet of roadway based on the desired scale. Then you would have to compute the stations at which the matchlines between sheets will occur. Next, you would draw the plotbox polylines in the plan view, following the roadway alignment and ending at these matchline stations. Then you would have to repeat this in the profile view. After you have drawn the plotboxes in both the plan and profile views, you could create dynaviews of these plotboxes in the sheet view in order to assemble a plan and profile drawing. Even though dynaviews provide a very powerful means of creating finished sheets that reliably illustrate the current state of the design, that's a lot of work. The ASAP module will do all that for you, and a lot more, in just a few seconds.

You control the entire process by editing the plan set configuration settings. If you wish to create alignment-based plans (such as plan and profile sheets), simply select the alignments for which you want plotboxes generated. **Terramodel** instantly creates both the plan and profile plotboxes. Below is the same subdivision illustrated in Figure 12-4, showing the plan view plotboxes for all of the roadways as created by **Terramodel**. You can imagine how much work just this feature alone could save you.

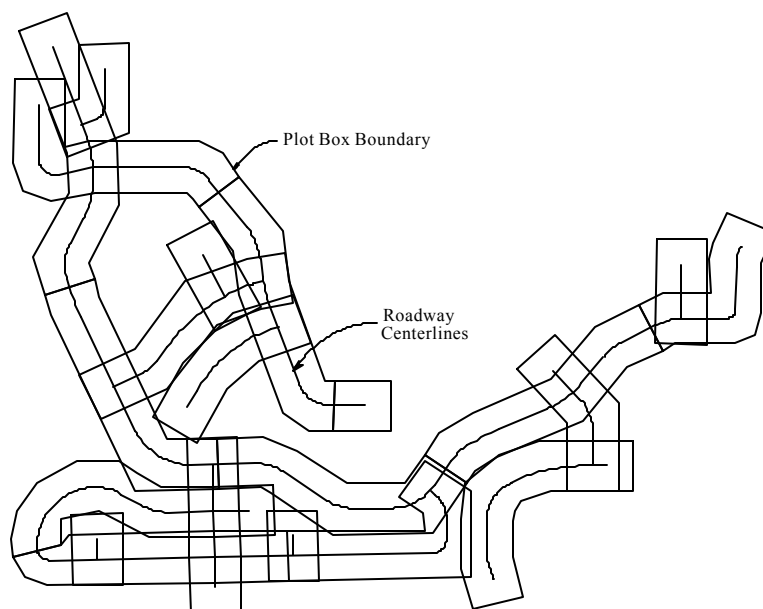


Figure 12-5 Typical Plan View Plotboxes for Subdivision

To create plotboxes, select the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on creating plotboxes, see **DRAFT Menu**, “Plotboxes (**psbox** command)—Alignment Plotbox Creation Command Bar”, in the on-line help system for the **psbox** command.

Editing Plan View Plotboxes

The type of plot box shape is determined by the plan set configuration settings, which you control. In a project such as in the above example, **Terramodel** determines the maximum length of roadway that can be shown on each sheet and establishes the matchline stations, based on the configuration settings.

Suppose a matchline placed in the default position falls in the middle of an intersection and you do not want the intersection to be divided between two sheets. You can either pick the matchline and drag it to where you want it, or enter an exact station to which it should be moved. **Terramodel** reconfigures both the plan and profile plotboxes on each side of the matchline to accommodate the move.

When you move a matchline, you will have shortened one plotbox while lengthening the other. Since they were presumably both at the maximum allowable length that will fit on a sheet, the lengthened one will now not fit properly. Simply instruct **Terramodel** to adjust the others, and every subsequent matchline along the alignment is adjusted (in both the plan and profile views) to compensate for the matchline relocation you made.

Plan view plotbox matchlines can be moved by selecting the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on moving a matchline between plotboxes, see **DRAFT Menu**, “Plotboxes (**psbox** command) — Move Alignment Plotbox Matchline Command Bar”, in the on-line help system for the **psbox** command.

The above discussion deals with matchlines between sheets. You can also have matchlines within one sheet. These are called *on-sheet matchlines*. They are useful when the curvature of an alignment results in a plotbox that will not fit properly within the allocated plan view area on the sheet. When this occurs, you can instruct **Terramodel** to break the plan view plotbox at a particular location (the on-sheet matchline), allowing the two sections to be rotated independently.

When you break a plan view plotbox into two plotboxes in this manner, **Terramodel** creates a dynview of each one on the sheet. It rotates them to better fit the plan view area, at the same time creating and labeling matchlines at the break (if desired), and placing a separate north arrow for each part. In the figure below, there are two plotboxes that potentially would require a break. The highlighted example is shown both in the overall plan view and as it might appear on a plan and profile sheet after rotating each part separately.

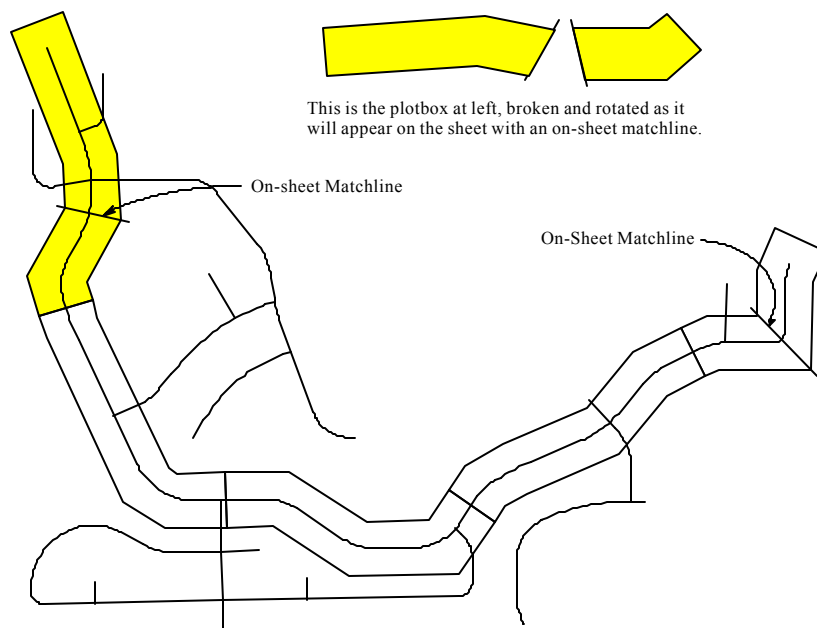


Figure 12-6 Breaking a Plan View Plotbox to Create an On-sheet Matchline

If you do not like the result, **Terramodel** can reassemble the broken plotboxes back into one, and update the sheet accordingly.



You can use Terramodel's polyline editing commands to achieve the exact plotbox shape you desire, if the default plotbox shape does not exactly meet your needs. The only restriction is that the matchlines must be perpendicular to the alignment.

Plan view plotboxes can be broken and rejoined by selecting the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on breaking plan view plotboxes to create an on-sheet matchline, see **DRAFT Menu**, "Plotboxes

(**psbox** command) —Break Alignment Plotbox Command Bar”, in the on-line help system for the **psbox** command. For more information on joining two broken plotboxes back into one, see **DRAFT Menu**, “Plotboxes (**psbox** command) —Join Alignment Plotbox Command Bar”, in the on-line help system for the **psbox** command.

Editing Profile View Plotboxes

As **Terramodel** creates a profile view plotbox, it attempts to position the plotbox vertically in order to best present the subject matter within the profile view of the drawing. You can specify a minimum and maximum height for the profile view in the plan set configuration settings. If there is little elevation variation in the subject matter, **Terramodel** will tend to center the subject matter in the view. If the elevations vary so that the subject matter will not fit within the minimum profile view height, the plotbox height is increased as needed (up to the maximum height that you specify). Such height increases are made in increments equal to one major horizontal grid line.

You can also set the elevation intervals for which the major and minor horizontal grid lines are drawn in the profile view. The profile view’s top and bottom elevations are constrained to coincide with a major horizontal grid interval.

Space limitations on the sheet obviously limit the maximum height of the profile view. You may, in fact, wish to restrict the maximum height to be equal to the minimum height—in effect specifying an exact and non-adjustable profile grid height.

You may encounter instances where steep terrain causes the variation in elevation of the subject matter within one sheet to exceed the range allowed by the maximum height. In such cases, **Terramodel** automatically breaks the content of the page into segments. It then shifts the profile grid’s elevation scale for each segment, as necessary, in order to keep the subject matter within the confines of the sheet’s profile view. Up to six such vertically shifted segments can be accommodated on each profile within a sheet.

While many of the **Planset** features act to lay out your drawings automatically, you have complete control over how these features are applied. You can change the height of a profile plotbox by one major horizontal grid line interval with the click of a button. Similarly, you can move the plotbox, or one of several vertically shifted plotbox segments, up or down by one major grid interval, thereby shifting the position of the subject matter within the sheet’s profile view.

In cases where vertically shifted profile plotboxes are used to accommodate steep terrain, the shifts always occur at one of the vertical grid lines in the profile. You can specify the stationing interval at which the vertical grid lines are drawn. Further, you can change the station at which a vertical shift occurs, moving it by one vertical grid line interval at a time.

You also have the option of inserting a vertical shift in the profile view. You might want to do this in order to make room for a note, for example. A simple button click will break the profile plotbox or segment into two segments. Each new segment can then be shifted vertically as needed.

Profile view plotboxes can be edited by selecting the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on editing profile plotboxes, see **DRAFT Menu**, “Plotboxes”, in the on-line help system for the **psbox** command.

Controlling Plotbox Visibility

You may have created plotboxes within the plan, profile or cross-section views for a number of sheet types. However, only those associated with the current sheet type are displayed at any one time. To see the plotboxes associated with another sheet type, you must select that as the current sheet type.

In a complex project, even though only those plotboxes associated with one sheet type are displayed, they may become very numerous and may overlap to the point where they create visual confusion, such as in Figure 12–5, above. If you wish to view only the plotboxes associated with one particular alignment, you can instruct **Terramodel** to isolate that alignment. You can also choose to hide all plotboxes. This is useful after creating your sheets, so that the plotboxes do not show up as borders within the dynaviews, though they can be excluded by the use of layer lists as well. Another command is available to show all plotboxes, after all or some have been turned off.

Plotbox visibility can be controlled by selecting the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on controlling plotbox visibility, see **DRAFT Menu**, “Plotboxes (**psbox** command)—Plotbox Visibility”, in the on-line help system for the **psbox** command.

Deleting Plotboxes

You can delete the plotboxes created by the **Planset** command, if you wish. You should, however, always do so using the plotbox deletion facility provided by that command, as opposed to using **Terramodel**'s **Delete** command. You can delete all of the alignment-based plotboxes associated with a given alignment by selecting the alignment and clicking the Delete Selection button on the Alignment Plotbox Deletion command bar. You can also choose to delete all plotboxes. If you have created sheets that are dependent on those plotboxes, **Terramodel** will advise you, and will not allow the plotboxes to be deleted until the sheets have been deleted.

Deleting a plotbox that is being used as a view port by an existing sheet, using Terramodel's **Delete** command, can cause numerous problems within sheets that rely on that plotbox.

Plotboxes can be deleted by selecting the **Plan Set/Plotboxes** option (**psbox** command) from the **DRAFT** menu. For more information on deleting plotboxes, see **DRAFT Menu**, "Plotboxes (**psbox** command) —Alignment Plotbox Deletion Command Bar", in the on-line help system for the **psbox** command.

Creating Drawing Sheets

Before you can create a drawing sheet, you must select the desired sheet type, create and edit the plotboxes, and place the sheet insertion point marker. These steps are described above. When you are ready, select the **Plan Set/Sheet Assembly** option (**pssheet** command) from the **DRAFT** menu. Then select the **Create sheets** option and designate the plan view plotboxes for which **Terramodel** is to create drawing sheets. **Terramodel** will create one sheet for each plotbox or plan/profile plotbox combination.



See the "Combining Sheets" section, below, for a discussion of combining two or more short alignment-based sheets into one sheet containing multiple short alignments. This is useful where you may be dealing with short blocks or cul-de-sacs for instance, where more than one will fit on a sheet. Up to four short alignments can be placed on a single alignment-based sheet if space permits.

Terramodel normally creates a new sheet consisting of, at least, the following items:

1. An outline plotbox based on the size of the associated base sheet.
2. A dynaview of the master sheet, typically containing the drawing border and title block.
3. One or more plan view dynaviews as appropriate, each with an associated north arrow.
4. One or more profile view dynaviews as appropriate.

Items three and four, above, are not included in miscellaneous or blank sheets. Item four is not included in plan or double plan sheets. The following items are optional depending on your needs:

1. One or more dynaimages associated with the plan view dynaview to display a raster image as currently loaded within the Image Manager and displayed within the plan

view. A dynaimage is a copy of the source image that is typically and clipped and rotated, and registered to the dynaview.

2. A profile grid consisting of major and minor horizontal grid lines, vertical grid lines, station and elevation labels associated with the major grid lines, and station equation labels.
3. Matchlines and labels, including the adjacent sheet number or drawing ID and the matchline station for alignment-based plans.
4. The sheet title.
5. The alignment name, for alignment-based sheets.
6. The sheet number, sheet total number, and/or drawing ID.
7. Plan and profile view labels.
8. Horizontal and vertical scales.
9. Designed by, drawn by, checked by and approved by staff name labels.

Only items four, six and nine are available in miscellaneous or blank sheets. You can create a sheet by selecting the **Plan Set/Sheet Assembly** option (**pssheet** command) from the **DRAFT** menu. Additional information regarding the process of creating sheets can be found in **DRAFT Menu**, “Sheet Assembly”, in the on-line help system for the **pssheet** command.

Controlling Displayed Drawing Content

The drawing content displayed within the plan or profile dynaview on a sheet can be controlled by the layer list assigned to that dynaview. You can designate the layer list to be assigned to plan and profile dynaviews within the plan set configuration settings. When a layer list is assigned to a dynaview, it displays the objects that are on the member layers of that list, and only those objects. The display of objects within the dynaview is independent of the current visibility of those layers. Consequently, a plan view dynaview within a sheet will show the correct content, even if some of the layers it displays are turned off and not visible in the plan viewing mode. It will not show objects on layers that are not members of its layer list, even if they are visible in the plan viewing mode. Your drawings always display the correct content, regardless of the layer visibility status of the plan and profile viewing modes. See chapter 5, **Understanding Terramodel CAD**, “Working with Layer Lists”, for a more detailed explanation of the creation and uses of layer lists. For more information on assigning layer lists to plan and profile dynaviews created by **Planset**, see **DRAFT Menu**, “Settings—Layer Settings Dialog Box”, in the on-line help system for the **pssset** command.

Controlling the Display of Dynaimages

Dynaimages allow you to create sheets that display raster images, such as aerial photographs, as background images placed beneath the plan view. A dynaimage is a derived raster image that **Terramodel** creates, based on a source image that occurs within the limits of a dynaview's plotbox. It is displayed in relation to that dynaview, providing a raster background. The dynaimage is therefore typically a portion of the source image that has been clipped, either to the dynaview's extents, or within some other user defined boundary. The derived image is also scaled and rotated to match the dynaview's implied coordinate system. For more information on dynaimages, see the on-line help system for the **image** command.

Dynaimages can be created by the **image** command. However, in doing so, though they are properly registered to the dynaview when created, they are not attached to that dynaview. Consequently, as the dynaview is moved or rotated, a dynaimage created in that manner is not moved or rotated as required to stay properly registered. However, dynaimages created by the **Planset** command are registered with the dynaview as attached objects. This allows the dynaimage to move with the sheets as they move within the plan set and to move and rotate with the dynaview, as you move it using the facilities provided within the **Planset** command.

You can elect to have **Terramodel** create dynaimages as it creates the sheets, or you can add them or recreate them later. You can elect to have the dynaimages clipped to the extents of the dynaview, or you can specify that they be clipped to a rectangular region that you designate within the sheet. For more information on the settings that control those decisions, see the on-line help system for the **psset** command.

When you add or recreate dynaimages after a sheet has been created, you can elect to display the dynaimage either of three ways. It can be displayed within the extents as defined by the dynaview's plotbox. It can be displayed within the default rectangular region, as noted above, or within a designated closed polygon boundary. For more information on the process for adding or recreating a dynaimage in relation to the plan view dyanview on an existing sheet, see the on-line help system for the **pssheet** command.

Controlling Placement of Labels and Titles

In general, you can control the placement of drawing titles and labels using the plan set configuration settings. You can designate whether or not they are to be included and, if so, their position, orientation, text formatting characteristics and content.

General labels are typically positioned by specifying x and y offset values with reference to the lower left corner of the sheet. However, the x coordinate of certain labels can be specified as centered over the profile grid area. Profile grid station labels are positioned

with reference to the top or bottom of the grid, and elevation labels with reference to the ends of the grid. Matchline labels are both aligned with and offset by a specified distance from the matchline. For more information on the inclusion, formatting and placement of labels, see **DRAFT Menu**, “Settings—Sheet Labeling Dialog Box”, in the on-line help system for the **psset** command.

Controlling the Placement of Dynaviews

When creating alignment-based plan or plan and profile sheets, **Terramodel** will rotate the plan view while placing it on the sheet. The north arrow associated with each plan view’s dynaview will be rotated accordingly. The positions of the rotated plan and the profile views and the north arrow are controlled by the current configuration settings. When creating plan and profile sheets, you can configure **Planset** to place the plan view on top, or the profile view on top, as you wish. See **DRAFT Menu**, “Settings—Sheet Configuration Dialog Box”, in the on-line help system for the **psset** command, for more information on the placement of plan and profile views and north arrows.

Recreating Drawing Sheets

If your configuration settings change during the creation of a plan set, you may wish to recreate an existing sheet in order to apply the new settings. Be advised that, in recreating a sheet, any adjustments you may have made to the location of plan or profile views, labels, etc. will be lost, as these components will be relocated in accordance with the new configuration settings. Any additional objects you may have drawn on the original sheet after **Terramodel** created it will remain. However, they will no longer be properly registered with the plan or profile views, which may have moved or changed in shape.

To recreate a sheet, follow the same steps you would use in creating a new sheet, but select the plotbox of the sheet you wish to recreate. Whenever you create sheets, if you select a plotbox which is already used in an existing sheet, **Terramodel** will detect the existence of the sheet and advise you. At that point, you may elect to recreate the existing sheet.

The recreated sheet will retain its sheet number and its original position within the plan set—regardless of the position of the sheet insertion point marker at the time you recreate the sheet. In all other aspects, however, you will be creating a new sheet.

Creating Blank Sheets

Planset allows you to create blank sheets, which can be useful for a variety of purposes (see “Using Miscellaneous or Blank Sheets”, above). When created, a blank sheet appears the same as a miscellaneous sheet. Unlike a miscellaneous sheet, however, a blank sheet can be of any supported sheet type.

Each sheet has an attribute containing its sheet type. The behavior of some operations (such as the creation of an index of drawings) depends on the sheet type. For example, when creating the index of drawings or the pop-up list of drawings that allows you to select a drawing by name, **Terramodel** can recognize sheets that contain alignment-based plans. The alignment names and stationing limits for these sheets can be added to the index.

When you create a blank sheet, you will be asked to explicitly designate its sheet type. That designation will over-ride the current sheet type setting. If the current sheet type is *plan and profile*, and you wish to create a blank *miscellaneous* type sheet, you do not need to change the current sheet type to miscellaneous. You'll just designate it as a miscellaneous sheet when it is created. Similarly, you can create a blank sheet of the plan and profile type, even when that is not the current sheet type. For example, you might have created a plan and profile sheet containing plan and profile drawings of three short cul-de-sacs, when a design change extends one of these streets by 120 feet. The three drawings no longer fit well on one sheet. You can create a blank single plan and profile sheet and then combine the crowded sheet with the blank sheet. By moving one of those roadways onto the blank sheet, you can leave the other two remaining with room to spare. By designating the blank sheet as being of the plan and profile type, it will behave like a plan and profile sheet.

You can create a blank sheet by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the button on the command line labeled Blank. See **DRAFT Menu**, “Plan Set Management”, in the on-line help system for the **psman** command, for more information on the creation of blank sheets.

Selecting Sheets

When executing a command in which you must select a master sheet or drawing sheet, you can do so graphically by picking any point within the desired sheet. To do so, you must view the sheet in the sheet view mode. If you have dozens of sheets in your plan set, using the **All** command to display the entire set may be time-consuming and not particularly helpful. A better option may be to pick it by name from a pop-up *drawing list*. Use the **psman** command, *View* option, *Drawing list* button and **Terramodel** will display a list of drawings containing the sheet number, drawing ID, and title of each. For

alignment-based plans, the list will also include both the alignment name and the stationing limits of each alignment on the sheet.

Placing the Sheet Insertion Point Marker

When you wish to insert a new drawing sheet, place the sheet insertion point marker after the master sheet or drawing sheet that you wish your new drawing sheet to follow. If you insert a sheet between two existing sheets, the marker appears in the gap between the sheet you select and the one that follows it. Before inserting the new sheet, all subsequent sheets in the queue will be moved to the right one sheet to make room for it.

When you create a new master sheet, it is placed following the sheet queue in which the sheet insertion point marker is located. Any subsequent master sheets and their sheet queues are moved down one row to make room for the new master sheet.

The insertion point marker is a filled rectangle that expands to the height of the sheet and to the width of the separation between sheets. You should be able to see it clearly and easily, as long as the gap between sheets is big enough. You can control the size of the gap between sheets as via the configuration settings. The marker is displayed using the color assigned to the \$MASTERSHEET layer. You can change its color by changing the default color for that layer.

You can place or move the sheet insertion point marker by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the **Ins. Pt.** button on the command bar. Then simply pick a point within the master sheet or drawing sheet after which you wish to place the marker. For more information on the insertion point marker, see **DRAFT Menu**, “Plan Set Management—Insertion Point Marker Command Bar”, in the on-line help system for the **psman** command. To learn how to adjust the separation between sheets, see **DRAFT Menu**, “Settings—Sheet Configuration Dialog Box”, in the on-line help system for the **psset** command.

Adjusting Drawing Sheet Components

Careful use of the plan set configuration settings will generally produce drawing sheets that meet your needs and do not require adjustment. However, there will inevitably be circumstances where you will want to make adjustments to the location of some objects. For example, you might wish to move the north arrow on a particularly full or complex sheet into an open area where it can be seen more easily. You can simply move the north arrow block to the position you prefer.

In general, feel free to move objects as you wish, but keep in mind the following cautions:

Avoid moving objects from one sheet to another. Most drawing sheet objects created by **Terramodel** have intelligence linking them to a particular sheet. If you move such an object to a different sheet, and then delete the original sheet, the linked object will be deleted as well. The one exception to this rule is in using the **Planset** combine sheet command to place some of the contents of one sheet on another where sufficient room exists. In that case, the link is altered to accommodate the move.

Do not use the Move or Rotate command to relocate a plan or profile view. Special **Planset** commands, as described above, are provided for that purpose. Their use assures that all attached and associated objects are moved as a unit.

Wait until you are satisfied with all other aspects of the drawing before making adjustments. If you make manual adjustments to a drawing sheet, the content of which is still likely to change, you may be creating extra work for yourself.

If you make adjustments which you later wish to undo, you can recreate the existing sheet in order to apply you default configuration settings to it. The sheet recreation process deletes most components of the sheet and recreates them, in which case the new objects will once again be located in their default position, undoing any adjustments that you may have made. This is also helpful if you need to update a sheet to reflect revised configuration settings.

Combining Sheets

When you create an alignment-based drawing, **Terramodel** automatically creates a separate sheet for each plotbox. Sometimes the last plotbox along an alignment is much shorter than the standard plotbox length, leaving you with empty space on a sheet. Short alignments, such as some cul-de-sacs, may also result in only partially-filled sheets. If you wish, you can combine the drawings from two separate sheets onto one sheet, in order to eliminate excessive white space. Up to four alignments can be combined onto one sheet, as space permits.

Combine sheets by selecting the “from” sheet and the “to” sheet. The leftmost alignment on the “from” sheet will be moved to the “to” sheet. If no drawings remain on the “from” sheet, you can delete the empty sheet.

You can combine two sheets by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the Combine button on the command line. For additional information on the combining of sheets see **DRAFT Menu**, “Plan Set Management—Combine Sheets Command Bar”, in the on-line help system for the **psman** command.

Reordering Sheets

You can change the order in which sheets are placed within the plan set. To do this, place the sheet insertion point marker where you wish to place the sheet(s) you intend to move. Then select the first and last sheet of the consecutive range of sheets you wish to move. To move only one sheet, select only the first sheet. To move all sheets associated with a given alignment, select any one, and then click the button labeled HAL. **Terramodel** will select the remainder of the associated sheets for you. Click OK, and the sheets will be moved to the place indicated by the insertion point marker.

As always, sheet numbers, textual references to sheet numbers, and the drawing index (if existing) will be updated to reflect the new drawing order. Note that when moving a sheet to a different sheet queue, it will take on the content of the master sheet associated with that sheet queue.

You can reorder the plan set by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the Reorder button on the command line. For additional information on changing the location and order of sheets within the plan set, see **DRAFT Menu**, “Plan Set Management—Reorder Sheets Command Bar”, in the on-line help system for the **psman** command.

You can also reorder entire sheet queues, moving the master sheet and all sheets in its sheet queue to another row, and adjusting the other sheet queues accordingly. This is accomplished by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the Master button on the command line, then selecting the Relocate queue command button.

Deleting a Sheet

You can delete a sheet or a range of consecutive sheets. To do so, you select the first and last sheet of the range of sheets you wish to delete. To delete only one sheet, select the first sheet only. To delete all sheets associated with a given alignment, select any one, and then click the button labeled HAL. **Terramodel** will select the remainder of the associated sheets for you. Click OK, and the sheets will be deleted.

Do not attempt to use Terramodel’s delete command to delete a sheet. You must use the sheet deletion facility within the **Planset** command in order to avoid damaging the integrity of the plan set. When plan set deletes a sheet, it moves the remainder of the sheets to the right in that queue into its place, and reconciles all of the sheet numbers and references to them. Numerous issues are reconciled by the **Planset** command, which can not be reconciled by the **delete** command.

To delete sheets from the plan set, select the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and click the Delete button on the command line. For more information on the deletion of sheets, see **DRAFT Menu**, “Plan Set Management—Delete Sheets Command Bar”, in the on-line help system for the **psman** command.

Viewing Individual Sheets and Sheet Information

In some projects the number of sheets can get quite large. To view a particular sheet, you need not execute the **All** command and then zoom in on the desired sheet. **Terramodel** will zoom the sheet view to the extents of any sheet you wish. You can select the sheet from the pop-up drawing list or pick it graphically in the sheet view. Then click the Zoom button to view the selected sheet. Buttons are then provided to zoom to the next or previous sheet.

In addition to viewing a sheet, you can get information on its content by clicking the Info button. The following data is presented in **Terramodel**'s report editor.

- Sheet number
- Drawing ID
- Sheet title
- Name and record number of the master sheet
- Name and record number of the base sheet
- Names and record numbers of the associated plan and profile plotboxes
- Alignment data if applicable:
 - Name
 - Record number
 - Station limits
 - Elevation limits
- List of sheets with notes referencing this sheet

This should give you all the information you need about a sheet. For example, if you are planning to delete a particular sheet, you can first determine if there are other sheets in the plan set that contain textual references to that sheet. This will allow you to remove any references or redirect them to a replacement sheet before deletion. (If a sheet with references to it is deleted, those references will display either a ? or the embedded

attribute text (EAT) code, depending on whether they were created by **Planset** or by the user, respectively.)

You can view or get information on a particular sheet by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the View button. See **DRAFT Menu**, “Plan Set Management—View Sheet Command Bar”, in the on-line help system for the **psman** command, for more information.

Plotting Sheets

You can easily plot individual sheets or ranges of sequential sheets from within the planset command. Once again you can select the sheets graphically, or from a pop-up drawing list which identifies them by sheet number, title, etc.

You can plot a particular sheet by selecting the **Plan Set/Plan Set Management** option (**psman** command) from the **DRAFT** menu, and clicking the Plot button. See **DRAFT Menu**, “Plan Set Management—Plot Sheet Command Bar”, in the on-line help system for the **psman** command, for more information.

Using the Project Data Editor

The remainder of this chapter describes how to use **Terramodel's** Project Data editor commands to maintain information about your project and to place references to that information in your drawings. The commands for this feature are located under the **DRAFT** menu.

The Project Data Editor is an important part **Terramodel's** ASAP module. As your project develops, you can use the Project Data Editor to enter information related to your project into a database which is stored in the **Terramodel** project file. There are two advantages to storing the project data in the project file. First, **Terramodel** can include it on the title page and individual page headings of reports which it generates. Second, and perhaps more important, you can link text objects anywhere in the project file to this project data. These text objects will always display the current value of the project data. If any of this information changes, and it will, you need only edit it in the Project Data Editor. Every instance of linked text throughout your entire plan set will be updated to display the revision, the next time **Terramodel** redraws the screen.

You can even link text objects to project data fields which are currently blank. In those cases the text object will display only an indicator of the project data field to which it is linked. When appropriate information is later entered in the editor, the associated text object will display it.

The Project Data Editor currently supports four general types of project information which will be of use to most users:

- General information
- Project Location
- Client data
- Staff data

The supported data is discussed in the following sections. The final section explains how you can customize the program to include revised or additional project data.

Editing Project Data

The Project Data Editor commands can be accessed from the menu bar within the DRAFT menu. Selecting **ASAP-Project Data** from the DRAFT menu displays a submenu containing the following five selections:

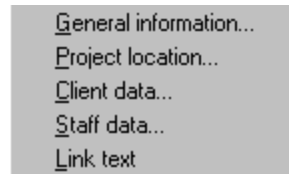


Figure 12-7 The Project Data Editor Submenu

Each of these commands are contained within a single TML command file. You can also use the **Projdata** command to open the Projdata command bar. Each command can then be selected from the command bar. Because these commands work together this is the preferred option. You of course also enter the command name from the command line.

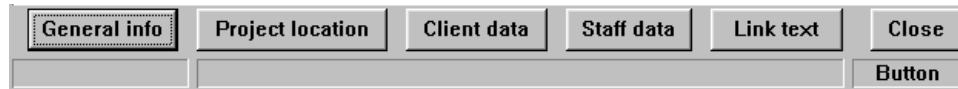
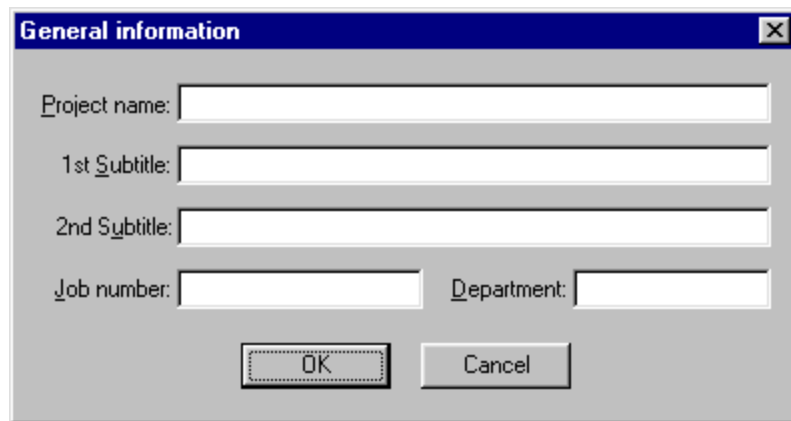


Figure 12-8 The Projdata Command Bar

General Information Dialog Box

Selecting **General Information** (**prjinfo** command) opens the *General Information* dialog box, shown below. It contains fields for the project name, two subtitles associated with the project name, the job number, and the department responsible for the project.



The image shows a standard Windows-style dialog box titled "General information". It features a blue title bar with a close button (X) in the top right corner. The main area is light gray and contains five text input fields. The first three fields are stacked vertically and labeled "Project name:", "1st Subtitle:", and "2nd Subtitle:". The last two fields are side-by-side, labeled "Job number:" and "Department:". At the bottom center, there are two buttons: "OK" and "Cancel".

Figure 12–9 The *General Information* Dialog Box

The maximum length of an entry is limited only by **Terramodel's** maximum text string length, which is 255 characters. The fields will scroll as needed to allow entry of text strings longer than the field width.

Project Location Dialog Box

Selecting Project Location (**prjloc** command) opens the *Project Location* dialog box:

The dialog box is titled "Project location" and contains the following fields and controls:

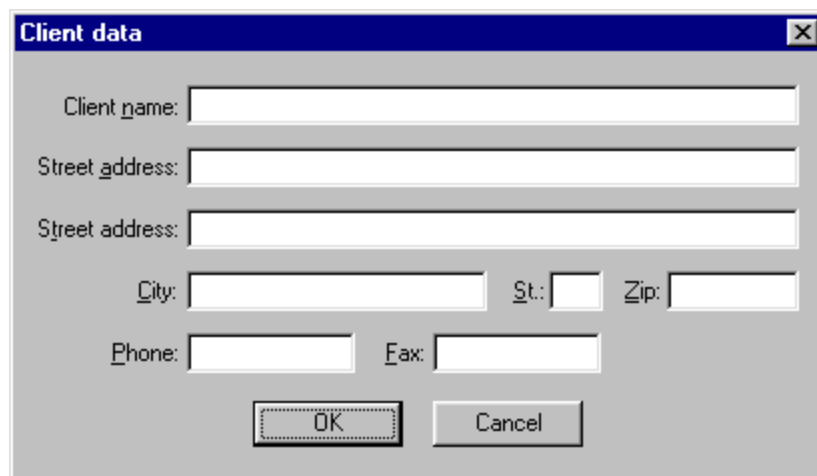
- Community: [Text Input Field]
- City: [Text Input Field]
- County: [Text Input Field]
- State: [Text Input Field]
- Section Number: [Text Input Field]
- Township Number: [Text Input Field]
- Township Direction: North (radio button), South (radio button, selected)
- Range Number: [Text Input Field]
- Range Direction: East (radio button, selected), West (radio button)
- OK button
- Cancel button

Figure 12–10 The Project Location Dialog Box

The community field allows you to designate the local community name in which the project resides. Additional fields are provide for the city, county, and state names. Should your project be located within an area where the U.S. Public Land Survey System is used, the section, township, and range numbers can be entered. The township and range directions are designated by selecting the appropriate radio button.

Client Data Dialog Box

Selecting Client Data (**prjclnt** command) opens the *Client Data* dialog box:



The image shows a screenshot of a Windows-style dialog box titled "Client data". The dialog box has a blue title bar with a close button (X) in the top right corner. The main area is light gray and contains several text input fields. The fields are arranged vertically: "Client name:" followed by a wide text box; "Street address:" followed by a wide text box; another "Street address:" followed by a wide text box; "City:" followed by a wide text box, "St.:" followed by a small text box, and "Zip:" followed by a wide text box; "Phone:" followed by a wide text box, and "Fax:" followed by a wide text box. At the bottom of the dialog box are two buttons: "OK" and "Cancel".

Figure 12–11 The Client Data Dialog Box

Fields are provided for your client's name, mailing address, phone number, and fax number.

Staff Data Dialog Box

Selecting **Staff Data** opens the *Staff Data* dialog box.

The screenshot shows a dialog box titled "Staff data" with a close button in the top right corner. The dialog contains the following fields:

- Project manager: [Text Box]
- Engineer of record:
 - Name: [Text Box]
 - Reg. no.: [Text Box]
 - State: [Dropdown]
- Surveyor of record:
 - Name: [Text Box]
 - Reg. no.: [Text Box]
 - State: [Dropdown]
- Designed by: [Text Box]
- Drawn by: [Text Box]
- Checked by: [Text Box]
- Approved by: [Text Box]

At the bottom of the dialog are two buttons: "OK" and "Cancel".

Figure 12–12 The Staff Data Dialog Box

This dialog box contains fields where you enter the project manager’s name; and the name, registration number, and state of registration of the engineer and surveyor of record. Additional fields are provided for the names of the persons responsible for design, drafting, checking, and approving of the plans.

The inclusion of an individual’s name in the project data implies project-wide scope. In other words, that the specified person will be responsible for checking or approving every sheet in the plan set. On larger projects, this may not be the case. When several people have worked on drafting the plans, the “drawn by” field in a drawing title block needs to be specific to each sheet. However, as design and drafting automation progresses, we are fast approaching a time when one individual can reasonably be expected to produce all of the drawings in a project. **Terramodel** will support your actual circumstances either way.

The ASAP module provides for the labeling of individual staff names for the “designed by”, “drawn by”, “checked by”, and “approved by” fields of a title block. If you need any of these to be specific to individual drawings, you should have **Planset** produce these labels on the particular sheets. If the responsible person (for any or all of these tasks) is consistent for every sheet in the plan set, however, place the text object linked to the appropriate project data field within the title block, as drawn on the base sheet. It will thereafter be displayed on every sheet in the plan set.

Linking Text Objects to Project Data

To link a text object to a particular project data field, select the **Project Data/Link Text** option from the **DRAFT** menu. After selecting the text object, click the button indicating which of the four information areas you are interested in. Below is the list box displayed when you click the Client Data button.

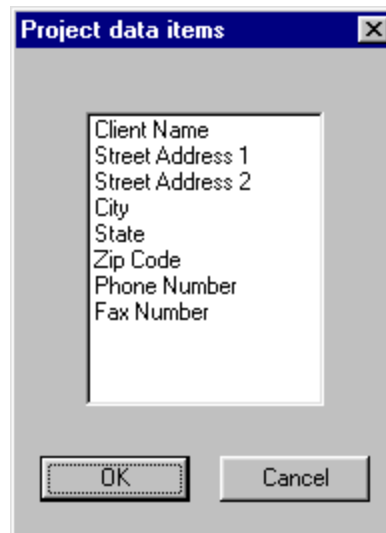


Figure 12–13 The Project Data Items List Box

Select from any of the displayed field names to assign that attribute to the text object. As you do, the text is updated to display the current contents of that project data field. For more information on linking text objects to project data, see **DRAFT Menu**, “Link Text”, the **prjltxt** command, in the on-line help system.

Customizing the Project Data Content

The Project Data Editor, as shipped with **Terramodel**, contains a data set that is typical of most user's needs. However, some users may want to augment or revise the data that it supports. For this reason the Project Data Editor is written as a TML command using the **Terramodel** Macro Language, and is distributed in source code form. It is actually a simple program, and a good example of an application that a user may wish to modify (one of the primary reasons that Plus 3 develops code using TML).

Given access to a Windows resource editor and a basic knowledge of programming, users will be able to customize the Project Data Editor to suit their specific needs. Otherwise, Trimble. can produce customized versions for a nominal fee.

For those who wish to customize the program, the source code is contained within the file named `projdata.tml`. The related Windows resource code is located within the file `tmodtml.rc`, and the associated attribute definition file is `projdata.adf`.

Information on programming with TML is included in the on-line TML help file, `tml.hlp`.

Chapter Thirteen

Using Coordcon

Terramodel Coordinate Conversion Routines

While in the past land surveyors had to occasionally compute and deal with state plane coordinates, today's surveyor is increasingly having to compute state plane coordinates for design or control points, as well as their transformations from an older datum to a newer one. Also, large survey projects may encompass multiple state plane grid zones, or the surveyor may be operating in an area that uses two or more overlapping zones. The land surveyor may also have to balance his raw field traverse data between control points that have known state plane coordinates.

Two forces are providing the impetus for this increased emphasis on state plane coordinates. The first is the drive toward GIS (Geographic Information Systems) by local, state, and Federal governments and utility companies, as they attempt to integrate their maps with operating data bases to produce automated land information systems. In the past, surveyors have used their own arbitrary coordinate systems for a majority of their individual land surveys. These independent surveys must now be placed on a single coordinate system in order to be used by a GIS system as a single overall map database.

The second driving force is the advent of Global Positioning Systems (GPS), increasingly used by surveyors to position and locate their traverse control points. These GPS systems normally produce latitude and longitude output, as well as state plane coordinates. Thus, the surveyor now has the ability to easily place all of his surveys on the state plane coordinate system of his choice. In response to this need, Trimble has added coordinate conversion capability to **Terramodel** to address these requirements

and to make it easier for the surveyor to convert and manipulate coordinate data as required.

If you are already familiar with state plane coordinate systems or grids, see "How to Convert Coordinates Using Coordcon", on page 505, for instructions on how to use the **Coordcon** command. The "Glossary of Geodetic Terms" on page 516 contains definitions which will help you understand the terminology in this chapter.

Uses for Coordcon in Terramodel

Coordcon lets you enter a latitude, longitude pair for a point or group of points and converts them to state plane coordinates or vice/versa. For selected points you can produce a listing of state plane coordinates, latitude, longitude, grid and convergence factors. You can also convert between state plane coordinates and the North American Datum of 1927 (NAD-27) or the North American Datum of 1983 (NAD-83) datums. You can also list a latitude and longitude for WGS84 datum.

Coordcon can assist you in creating map overlays and joining together maps that are geographically adjacent but whose coordinate systems are disjointed. Use **Coordcon** to convert between coordinate systems based on these two datums, between State Plane zones, and between UTM zones.

Coordcon defines the following map projections:

Albers Equal Area	
American Polyconic	Azimuthal Equidistant
Bipolar Oblique Conformal Conic	Bonne
Cassini	Eckert Iv
Eckert Vi	Equal Area (Authalic) Cylindrical, Normal Aspect
Equal Area (Authalic) Cylindrical, Transverse Aspect	Equidistant Conic
Equidistant Cylindrical	Gnomonic
Goode Homolosine	Hotine Oblique Mercator
Lambert Conformal Conic	Lambert Equal Area Azimuthal
Lambert Equidistant Azimuthal	Lambert Tangential
Mercator	Miller Cylindrical
Modified Polyconic	Modified Sterographic
Mollweide	New Zealand National Grid
Orthographic	Robinson Cylindrical
Stereographic	Transverse Mercator
Unity Projection	Vandergrinten

Map Projections and Coordinate Transformations

Imagine that your task is to compute the area of the state of Wyoming and provide its dimensions in terms of degrees of latitude and longitude. You could calculate an approximate answer in so many square degrees, but does that answer have any meaning? The answer is, of course, no because the distance between degrees of longitude varies over the surface of the earth. Stating the dimension of a line in degrees of longitude gives no idea of the true length of that line.

A better system uses cartesian coordinates, where the distance between one unit of X is the same as the distance between one unit of Y. Such a system usually calls for X coordinates (eastings) along a horizontal axis and Y coordinates (northings) along a vertical axis. Thus, we need a method of translation from the curved surface of the earth

to a cartesian coordinate system that is generally used in **Terramodel** survey or design projects.

Everyone is aware that the earth is round. However, when a surveyor or civil engineer produces a plat or a plan, the plan is flat. All surveying, engineering design, GIS systems, and CAD systems produce flat maps. This work usually consists of design, drafting, or topographic drawing in cartesian coordinates. For small projects such as subdivisions and site designs, this is fine. However, when your project encompasses a very large area (for instance, a state or worse yet, the entire United States), the roundness of the earth will introduce distortion. Think, for example, of a tennis ball or an orange peel. If you were to try to cut either of these up and make them flat, you could only do this if you stretched or deformed them or cut them into very small pieces.

Producing maps (versus the plans and plats of engineers or surveyors) of large geographic areas where curvature becomes a factor has primarily been the province of cartographers (map makers). Cartographers must convert the linear cartesian coordinates into angular spherical coordinates of latitude and longitude when they want to use the surveyors' or engineers' plats to build an overall map for a very large area without introducing distortion.

Both mathematicians and cartographers have struggled for more than 2,000 years to represent the spherical earth on a flat piece of paper. Mapping even portions of the spherical earth on a computer display or plotter causes similar problems. Two transformations are actually necessary. The first calls for a scale reduction in order to fit such a large area on a sheet of paper. The second transformation deforms the rounded surface of the earth to make it flat.

While there are numerous ways to derive coordinate systems or map projections that will represent the spherical earth on a flat piece of paper, it cannot be done without incurring at least some distortion in the results. The goal when choosing the proper coordinate system is to minimize the distortion of the mapped area. In the following discussion, you will see that some projections have standard lines (that do not change in length when projected). Others will have a uniform scale in only one direction (equidistant) while others maintain directions with respect to a central point (azimuthal). Some projections are called equal-area, wherein the relative size of all features projected is maintained. Other projections are conformal, which means that they will have equal scale factors in all directions at any one point. However, no map projection can be both equal-area and conformal. We will see that there are many variations and combinations of projections, but they all must sacrifice something due to the distortion induced in going from a curved surface to a flat sheet of paper or a computer screen.

Should there be any doubt about the need for the cartographic process, consider the map of the United States (shown in Figure 13-1) which was plotted using latitude and longitude values as cartesian X and Y coordinates.



Figure 13-1 Unprojected Map of the USA

While the features of the map are recognizable, certainly you would not want to make any meaningful decisions based on information obtained from this map. For example, would you get on an airplane if you knew the captain used a map such as this to estimate the flight distance and from this, how much fuel to take on for the flight?

While there are situations in which latitude and longitude coordinates are used often (navigational and GPS systems, for example), the analysis, interpretation, and decision-making processes for engineering, land surveying, and GIS projects require that the spherical geographic data be flattened properly for placement on paper by a cartographic process. That is, we prefer maps that look more like that shown in Figure 13-2:



Figure 13-2 Conic Projection of the USA

The Latitude and Longitude Coordinate System

Before we go any further, let's briefly review what latitude and longitude coordinate systems are. Latitude/longitude systems are non-cartesian (i.e., they are spherical) and must be converted to a cartesian system for adequate engineering or survey mapping. Examine Figure 13-1 again to see how squashed the U.S. appears when latitude and longitude are treated as simple X and Y cartesian values.

Parallels of latitude are evenly spaced east-west imaginary lines that circle the earth and are parallel to the equator. Latitude is measured in degrees north and south of the equator to either pole. The longest parallel is, of course, the equator which is at zero degrees. The parallels increase in angular designation until, at the north and south poles, the parallels are no longer circles but points, with an angle of 90 degrees from the equator. Technically speaking, the geodetic latitude of a point is the angle that a line normal to the spheroid (a mathematical model of the earth) makes with the plane of the equator.

Meridians of longitude are imaginary north-south lines that intersect at the north and south poles of the earth. The distance between these lines is greatest at the equator and shrinks to zero at either pole. Meridians of longitude intersect parallels of latitude at right angles. Longitude is measured in degrees east or west from a starting point known as the Prime Meridian. While there have been many prime meridians over the course of history in France, London, and Philadelphia, today's prime meridian was chosen somewhat arbitrarily. It was adopted at the International Meridian Conference of 1884 as the north-south line "passing through the center of the transit instrument at the Observatory of Greenwich, England as the initial meridian for longitude". The primary reason for choosing this meridian was that the world was relying heavily on British nautical charts at the time for navigation and those charts used this meridian.

Figure 13-3 shows a representation of latitudinal and longitudinal lines:

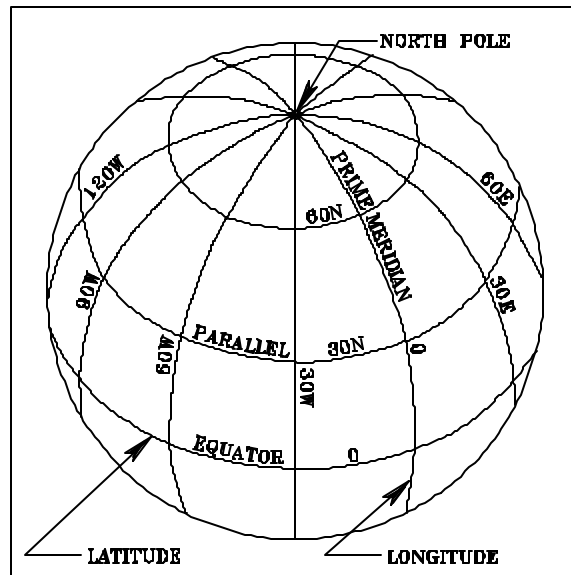


Figure 13-3 Latitude (Parallels) and Longitude (Meridians)

The location of any point on the surface of the earth can be described in terms of the intersection of the arcs of latitude and longitude that we have described. These locations on the earth are usually described by a coordinate pair, for example: Latitude 45° 56' 40.2" N, Longitude 119° 18' 01.3" W describes one unique point on the earth that happens to be in the state of Washington. The network of meridians and parallels in 13-4 is often called a "graticule" of the earth which, as we will see, can become a reference network for all points on the globe, just as if it were the earth itself.

The method of determining latitude and longitude of a given point on the earth has an interesting history. In practice, latitude can easily be determined. Early Polynesian sailors traveling from Tahiti to the Hawaiian Islands could determine their latitude simply by observing the angle made between the north pole star Polaris, the observer, and the horizon in the northern hemisphere. Polaris is peculiarly situated almost directly over the north pole, along a vertical extension of the earth's axis. At the north pole, Polaris is fixed directly overhead, and the equator is fixed on the horizon. Consequently, Polaris can be used to determine latitude in the northern hemisphere and also as an accurate navigational beacon.

Longitude, however, is considerably more difficult to determine by observation. The inability to determine longitude was a serious hazard to early ship navigation. The Spanish government in the mid-1600's offered huge sums of money to the inventor who could come up with a reliable way to measure longitude to help prevent shipwrecks of

valuable treasure. However, the development of such a method was dependent upon accurate time-keeping ability.

In 1714, John Harrison, an Englishman, developed the marine chronometer. This device keeps precise time. When the sun passes directly over the ship, the time is noted and compared with the time that the sun passes directly overhead in Greenwich. By using correction tables, the local longitude can be calculated. It is also possible to make sun or solar shots at other times of the day and apply additional correction factors to determine local longitude.

Today, advanced techniques in remote sensing using satellite mapping or GPS systems can accurately determine the latitude and longitude of any point on the surface of the earth. This new technology has made the observations recorded on most topographic maps obsolete. This is an issue to which we will return when we discuss the North American Datum of 1983 (NAD83).

Before we go on, there is one item we should discuss in regard to meridian lines that is called "convergence".

Convergence of Meridians

"Convergence" (sometimes called mapping angle) is a term that land surveyors must deal with when working with meridians on either state plane coordinate systems or U.S. Public land surveys. Simply defined, convergence is the angular difference between a geodetic azimuth and the grid azimuth. However, when traverse lines exceed 5 miles a second term, or refining factor, must be included. (The computed convergence angles in **Terramodel** include this refining factor for maximum accuracy.) Grid meridians are parallel, whereas geodetic meridians converge as they run northward. Normally surveyors tie to grid azimuths at each end of a long survey, so it does not have to be considered. However, if you want to take a Polaris or solar observation somewhere along the way, you must consider convergence. See 13-4 for a graphic representation of a convergence or mapping angle.

In a Lambert State Plane Coordinate System Zone, convergence varies uniformly with longitude. The rate of change of the convergence (or mapping angle) per unit of longitude is one of the constants in any given Lambert coordinate zone.

In a Transverse Mercator Zone, convergence is very nearly equal to the difference in longitude between a station and the central meridian times the sine of the latitude at the station. An additional term (that amounts to less than 1/2 second) is needed for maximum accuracy, and **Terramodel** accounts for this, also.

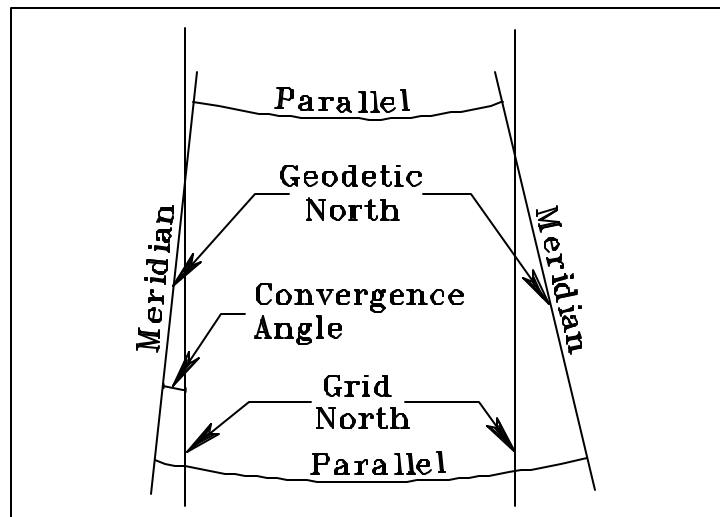


Figure13-4 Convergence or Mapping Angle

You can obtain the convergence (or mapping angle) of any grid system coordinate point in **Terramodel** by using the **List** option of the **Coordcon** command.

Geodetic Control Networks and Earth Datums

Geodesy is a rather imposing term, but broken down into its roots, it simply means the "Study of the division of the earth". The term specifically refers to the science of measuring the size and shape of the earth and determining the exact positions of points on its surface. The National Geodetic Survey is the Federal agency in charge of collecting and archiving benchmark data for geodetic control networks, restoring obliterated benchmarks, and providing information and services to both industry and the public.

A geodetic control network is a set of points with coordinates that are consistently interrelated in some predictable and documented way. The precision of each point is categorized into accuracy orders and classes. Each accuracy order is a measure of relative positional error for points in the network, or the degree to which a particular point can be trusted. A measurement of 1:100,000 indicates a minimum distance accuracy of 1 unit per every 100,000 units measured relative to other points in the network. The most common orders and classes of accuracy are as follows:

Table 13-1 Orders and Classes of Accuracy

Classification	Minimum Distance Accuracy
First Order	1:100,000
Second Order, Class-I	1:50,000
Second Order, Class-II	1:20,000
Third Order, Class I	1:10,000
Third Order, Class-II	1:5,000

A map datum is the mathematical surface passing through these points. A geodetic control network, by contrast, is a system of man-made monuments that are referenced to this datum surface. A geodetic control network's purpose is to provide a reference framework for the systems of existing independent surveys. Individual points are sometimes called "benchmarks", because they can be used to benchmark or tie other surveys together in a common net or more likely, become the starting point for a new survey.

Map Datums and How They Are Created

A map datum (for land observation-based ellipsoid systems) is established by taking a given reference ellipsoid (which specifies the size and shape of the earth mathematically) and a base point on the earth's surface from which the latitude and longitude and all other coordinate points are referenced. This point is then assigned a latitude and longitude, an elevation above the ellipsoid, and an azimuth to a known point. Remember, this origin or surface base point only exists in land observation-based datums such as the North American Datum of 1927 (NAD-27). Many ellipsoids of the earth have been developed and used, so we will next explore their use.

The Earth As An Ellipsoid

Many maps, including most commercial world atlases, assume that the earth is a sphere or a globe that is perfectly round. In 1687, Isaac Newton was the first to determine mathematically that the earth ought to be slightly flattened at its poles due to rotational forces. The French, however, believed the earth to be egg-shaped, based on meridian measurements they had made in France. To settle the dispute, the French Academy of Sciences sent expeditions to Peru in 1735, and to Lapland in 1736, to measure the distance between meridians at widely separated latitudes. Their measurements confirmed Newton's conclusions. These measurements led to many more meridian measurements. For example, between 1799 and 1951 there were 26 separate determinations of the dimensions of the earth, most with different answers.

With today's technology, geodesists have found that the radius of the earth at the equator is greater than the radius at the poles by about 21.4 km or 13.3 miles. This is about 1 part in 300. The resulting shape is called an oblate ellipsoid of revolution or an oblate spheroid.

As we will be using it, the term ellipsoid is mathematical terminology for an equation approximating the shape of the earth. This ellipsoid fits the earth in most places within a few hundred meters. However, since the internal density of the earth is not as uniformly distributed as mathematical models, a "best-fitting" ellipsoid for any particular portion of the earth may not be "best-fitting" elsewhere on the earth. Thus, applying survey control networks to ellipsoids has resulted in the need for many different ellipsoids for "best-fits" on various parts of the earth.

Defining the ellipsoid is just one step toward building a coordinate or map projection system tied to the earth. Since all map projections from the earth require an ellipsoid definition, we will explore the process further.

Calculating an ellipsoid is an arduous task for ground-based observers. Consequently, various cartographers over the centuries have found different values for the ellipsoid by measuring the polar and equatorial radii of the earth. Early ellipsoids, such as the Clarke ellipsoid of 1866, were calculated for relatively small areas of the earth (such as North and Central America). In addition, other ellipsoids were also developed for local parts of the world, as shown below:

Table 13-2 Early Ellipsoids for the Earth

Ellipsoid Name	Date Developed	Used Where
Airy and Airy ⁴⁹	1830 & 1849	Great Britain
Everest	1830	India & Surrounding Area
Bessel	1841	China, Japan, W. Germany, Chile, Indonesia; Central Europe
Clarke	1866	North & Central America; and the Philippines
Modified Clarke-IGN	1880	Africa and France
Hayford	1909	Remainder of world
International	1924	Europe & Remainder of world
Krasovskiy	1942	USSR
Australian National	1965	Australia
South American	1969	South America

Because of the use of ground-based observations, these early ellipsoids were not precisely earth-centered. Extending 19th century ellipsoids to the entire earth further results in undesirable errors outside local areas of interest. World-wide ellipsoids were just impractical until the advent of satellite measurements.

The Clarke 1866 ellipsoid is a measurement that has been used widely for mapping North and Central America since 1927, when NAD-27 was established. Geodesists such as Clarke were responsible for more than one ellipsoid, so it is important to note in which year the ellipsoid was published. For example, Clarke published ellipsoids in 1858, 1866, and 1880.

The recent advances in satellite geodesy have resulted in the Geodetic Reference System Ellipsoid of 1980 (GRS80). This reference ellipsoid is earth-centered and is the basis for the North American Datum of 1983 that we will discuss in the next section. Another almost identical, earth-centered ellipsoid was developed by the Army and is called the World Geodetic System of 1984 (WGS1984). WGS84 is also in widespread use and the differences between the two ellipsoids are quite trivial, as shown below:

Table 13-3 Selected Ellipsoid Dimensions

Ellipsoid Name	Date Developed	Equatorial Radius	Polar Radius
CLARKE 66	1866	6,378,206.4 m	6,356,583.8 m
GRS 80	1980	6,378,137 m	6,356,752.3141 m
WGS 84	1984	6,378,137 m	6,356,752.3142 m

The choice of an earth-centered ellipsoid is a dramatic departure from older systems. Note that the position of the radius is definitely not the same in Figure 13-5:

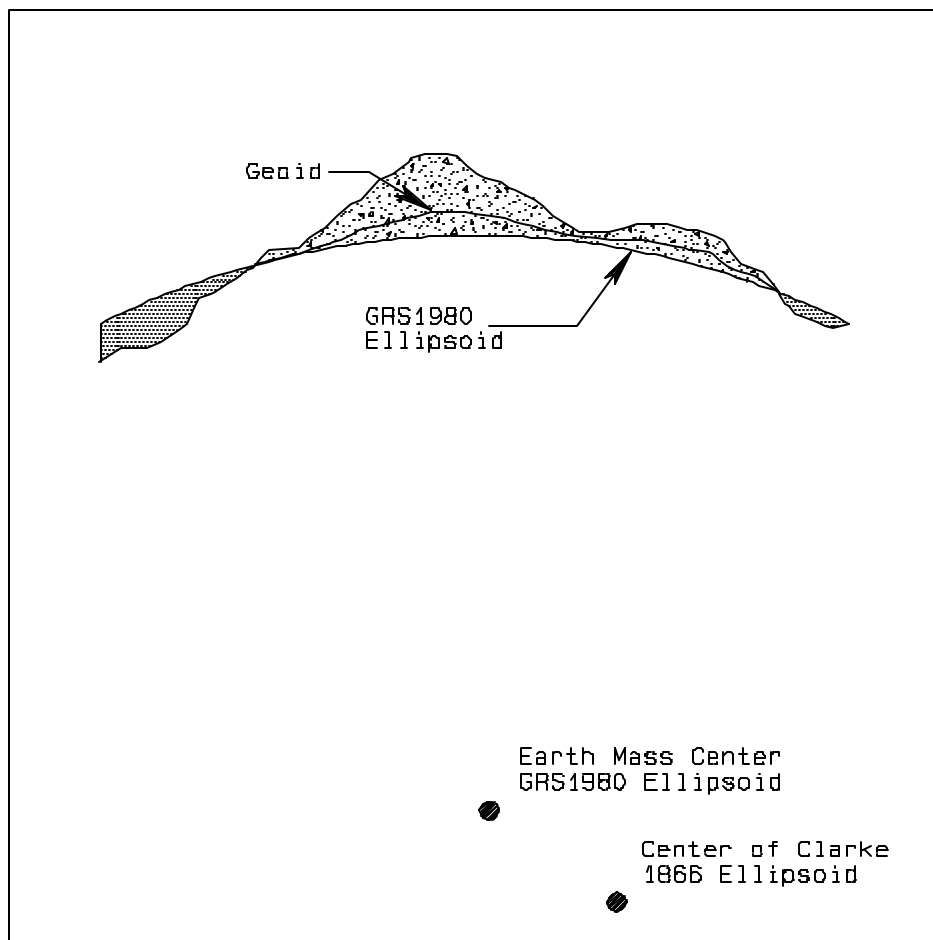


Figure 13-5 Clarke 1866 and Global Reference System of 1980 Ellipsoids
(Adapted from the publication *State Plane Coordinate System of 1983* by National Geodetic Survey)

Figure 13-5 shows the relationship between the Clarke 1866 and Global Reference System of 1980 ellipsoids. Only the center of the Clarke ellipsoid is shown.

You may also have noted that there is a "Geoid" line denoted in Figure 13-5. Although you can define an ellipsoid for the earth, the earth's surface will actually deviate somewhat from the mathematical figure. The "geoid" is the *mean sea level surface* used for most vertical elevation measurements such as contours. In other words, "geoid" is the name given to the shape that the earth would assume if it were all measured at mean sea level. As shown in Figure 13-4, the geoid is an undulating surface that generally varies by not more than about eighty (80) meters above or sixty (60) meters below an ellipsoid

that fits the earth well. This variation is far less than the amount that the ellipsoid varies from a true sphere.

It is important to note that elevations and contour lines on the earth are reported relative to the geoid and not the ellipsoid. On the other hand, latitude, longitude, and all state plane coordinate systems are determined with respect to the ellipsoid. Make sure when you are dealing with elevations that you know whether they are referenced to the ellipsoid or to the geoid.

A direct effect of using different ellipsoids is that the latitude/longitude coordinates for ground points that were assigned under one ellipsoid change under the new ellipsoid. The ramifications of the adjustments called for by the 1983 system (NAD-83) are only now becoming apparent. For example, in some areas the latitude and longitude coordinates can vary by up to 300 meters between NAD-27 and NAD-83.

Ellipsoid Equatorial and Polar Radii

As you may have noted in Table 13-3, two measurements are sufficient to calculate an ellipsoid. These measurements are the equatorial radius (or semimajor axis) and the polar radius (or semiminor axis). The equatorial radius is the distance from the center of the earth to the ellipsoid surface as measured along the plane of the equator. Likewise, the polar radius is the mean distance from the center of the earth to the North and South poles, measured normal to the plane of the equator.

Ellipsoid Flattening

As we noted previously, there is a flattening of the earth (at the poles) due to its rotation. This is generally called "flattening" and is a measure of the deviation of the earth from a perfect sphere. All rotating planetary bodies experience flattening to some degree. The flattening is an intermediate result in the calculation of the eccentricity of the ellipsoid. This result is ultimately used by the projection equations that we will be discussing later.

Geodesists calculate the amount of flattening by using the following equation, where f is the flattening, r_p is the polar radius, and r_e is the equatorial radius:

$$f = 1 - (r_p / r_e)$$

Often, the flattening will be represented in tables as an integer fraction where the numerator is unity and the denominator is an integer.

Ellipsoid Eccentricity

Once the flattening factor (f) is determined, most map projections use some form of eccentricity, rather than the flattening factor itself. Eccentricity is calculated using the following formula:

$$e^2 = 2f - f^2$$

The value of eccentricity for the GRS80 ellipsoid (e^2) is 0.0066943800 and for the Clarke 1866, it is 0.006768658.

United States Ellipsoids and Datums

Ideally, geodetic control networks should exist before the land surveys, but generally, history has not allowed this. Such was the case when the United States Datum of 1901 was established, which was used to tie together many surveys in New England. This datum had an origin in Principio, Maryland, and was in use until a formal readjustment of the national network, the North American Datum of 1927 (NAD-27) became available.

The North American Datum of 1927 (NAD-27)

By the mid 1920's the problems of trying to tie new surveys into the 1901 datum were becoming serious. In response the U.S. National Ocean Survey established the North American Datum of 1927 (commonly shortened to NAD-27). This datum used the Clarke ellipsoid of 1866. The datum origin station is located at Meades Ranch in Kansas and was used to determine the latitude and longitude of over 50,000 surveying monuments throughout the contiguous United States and Alaska. The monuments served as starting points for many more local surveying and mapping projects. Over the years, stations were added to the network and until there were over 267,000 stations of varying quality that were part of NAD-27.

NAD-27 was a big improvement over the United States Datum of 1901, but it was still earth-based, (since it had an origin at Meades Ranch in Kansas). Since NAD-27 was earth-based, its accuracy and precision deteriorated rapidly as one moved away from Kansas. For example, the Alaskan Highway was defined by a single traverse that connected Alaska with the lower 48 states. Obviously, since the traverse was a long distance from Kansas, its computed geodetic coordinates were not very accurate.

Not long after the establishment of NAD-27, it was recognized that NAD-27 was only at best, a stop gap measure. Many surveyors and geodesists began ignoring the national system, because it lacked sufficient control for their purposes. Considering the field equipment available at the time, the North American Datum of 1927 was a remarkable database. However, technology had improved greatly since that time, and the demands for greater precision and accuracy in surveys continued to increase. NAD-27 was insufficient to service the precise satellite and distance measurement information that was becoming available during the 1970's. Finally, in the late 1970's, a new datum using satellite measurements was proposed, and in 1983, a new datum having the center of mass of the earth as its origin was adopted. This new datum is called the North American Datum of 1983, or NAD-83 for short.

North American Datum of 1983 (NAD-83)

The North American Datum of 1983 (NAD-83) differs from the previous NAD-27 datum in several respects. In the first place, the calculated location of the center of the earth is different. NAD-27 used the Clarke 1866 ellipsoid, while NAD-83 uses the Geodetic Reference System ellipsoid of 1980 (GRS1980) as its center. Note that the term "geodetic reference system" is a synonym for the word datum. GRS 1980 was computed by using satellite data. The center of the satellite's orbit is by definition, the center of the earth's mass.

Secondly, this project conducted new surveys in areas of sparse control, and adjusted the latitude and longitude of the more than 257,000 survey stations (benchmarks) in the United States that were part of the older NAD-27. An equation with more than 500,000 unknown variables was solved, requiring 180 staff years to complete. The North American Datum of 1983 is thus a calculated and more precisely executed enhancement of the older but still valuable, existing NAD-27 data.

As we mentioned previously, changing the datum also changed the latitude and longitude values for most control points in North America, some by as much as 300 meters. This occurred when the origin of the datum was moved from Meades Ranch in Kansas (the earth-based origin for the NAD-27 datum) to the center of the earth's mass for the NAD-83 datum. Satellite measurements now make it possible to determine the precise latitude and longitude coordinates of most points on the surface of the earth.

NAD-83 solves several problems. NAD-27 map error is removed, satellite data can be more precisely overlaid, a new system of State Plane Coordinates based on the meter can be established (while at the same time correcting deficiencies such as negative coordinates in some areas), and map data can finally be integrated on a global scale with other countries who are also using an earth-centered ellipsoid.

In the past, most people in industry and government have been satisfied with coordinates that were correct relative to a specific survey or to other local boundaries. In some states, coordinates cannot even be entered into the legal record. However, the day of relying upon the relative accuracy of coordinates is passing. Given the new sources of GPS satellite data, it is both perilous and costly to rely on relative accuracy when absolute coordinates are available.

As computer mapping increases and databases grow, Geographic Information Systems (GIS) are becoming the norm for governments and local municipalities. The correctness of absolute coordinates will become ever more critical as this trend continues and these organizations try to accurately assemble maps for their facilities or land (that contain many different coordinate reference systems) and relate them directly to their operating databases or systems.

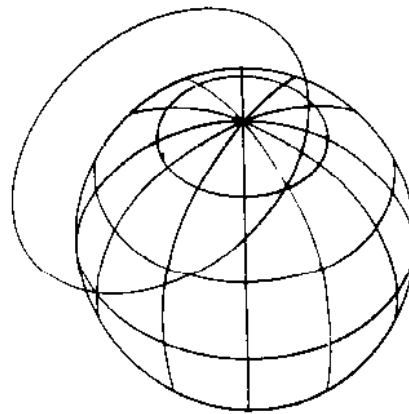
We have reviewed several components that are needed to produce state plane coordinates, i.e., ellipsoids, and datums (plus their resulting latitude and longitude), but one major piece is still missing. This missing piece is the concept of a map projection; to convert spherical data on the curved surface of the earth to a flat map or for use in **Terramodel**.

Map Projections

As we stated previously, engineering design and CAD computer systems such as **Terramodel** produce drawings that are not only flat, they are cartesian. In other words, these systems want X's and Y's where the X axis is orthogonal to the Y axis and, unless you are working in the profile view of **Terramodel**, every unit of X is the same as every other unit of X and Y. Generating a cartesian image of geographic data inside of **Terramodel** requires the same procedure as if you were committing it to a piece of paper, except we just omit the final step of plotting it out. In other words, a "projection" is involved in converting the "curved surface" data to a flat map or vice versa. However, there is a bit more to making a map than just a projection. What we require is a coordinate system definition which involves the use of a projection. We'll return to the concept of a cartographic coordinate system after we've taken a brief look at what we mean by the term "map projection".

Azimuthal Map projection

Imagine, if you will, the earth (and all of the features on the earth that are of interest to us) consisting of a wire mesh globe, and that a huge flash bulb is positioned at the center of the earth. To obtain an image of the round earth on a flat piece of photographic film, we could simply hold the film up to the surface of the globe and expose the film by strobing or flashing the light source, as depicted in Figure13-6:



**Oblique Azimuthal
(plane)**

Figure 13-6 Photographing an Oblique Azimuthal Plane

If we developed the exposed film after taking a picture of the previous plane surface, the resulting photograph might appear as depicted in Figure 13:

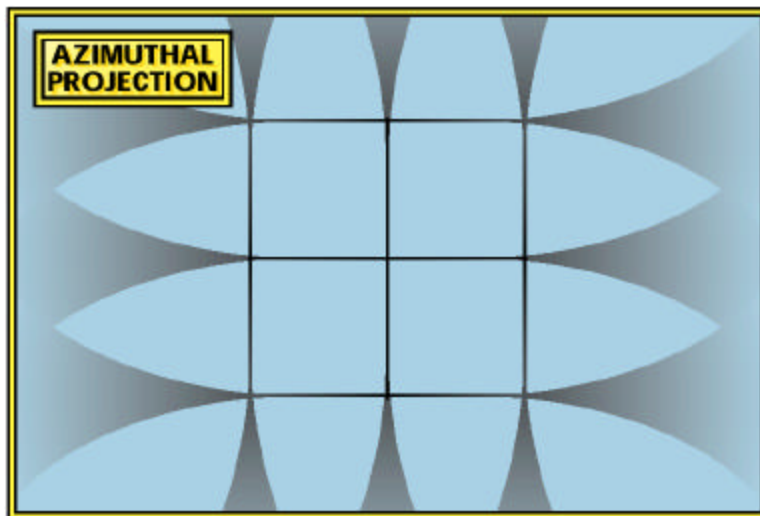


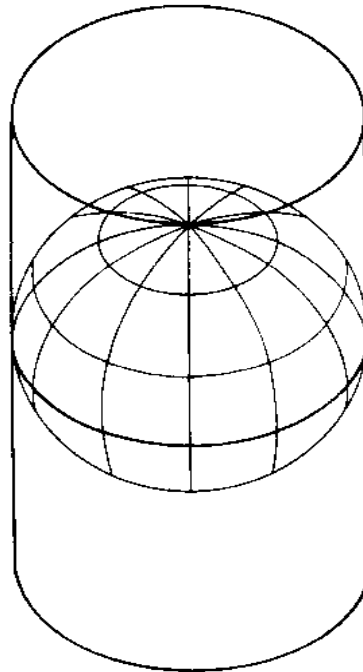
Figure 13 Photograph of Azimuthal Projection

It should not be any great surprise to anyone that the lines defining the features of the globe in Figure 13 are clean and crisp where the film was very close to the surface of the globe when we exposed the film. Where the film was a significant distance from the surface of the globe however, the lines on our photograph are fuzzy. As the distance between the photographic film and the surface of the globe increases, the fuzziness increases until we reach a point at which the lines completely disappear.

Conceptually, the process of projecting spherical geographic data onto a flat map is exactly the same. The technique that we have just visualized is essentially what happens mathematically when an Azimuthal projection is produced. In such a projection, the area about the origin of the projection (i.e. the point where the film actually touched the earth) is distortion free and the distortion introduced by the projection increases as one proceeds away from the origin (i.e., as the distance between the film and the globe increases).

Cylindrical (Mercator) Map Projection

You don't need to be a rocket scientist to figure out that you can get a better map by figuring out a way to get more of the photographic film closer to the surface we are mapping. Gerardus Mercator figured this out in 1569 and developed the idea of taking the film and wrapping it around the earth in the form of a cylinder. In so doing, the film actually touched the earth along the whole extent of the equator. We can use our imaginary apparatus to do the same thing (wrap the earth with film at the equator), as shown in Figure 13-7:



Regular Cylindrical

Figure 13-7 Mercator's Cylindrical Projection

If we then trigger our flash at the center of the earth and expose the film, we get a photograph that looks something like that shown in Figure 13-8:

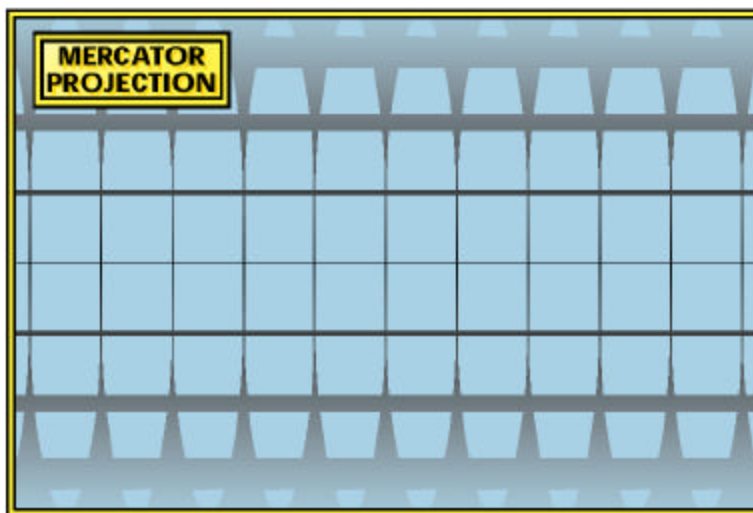


Figure 13-8 Photograph of Mercator's Projection

Notice that anywhere close to the equator the lines are crisp and clean, indicating a minimum of projection distortion. As we proceed away from the equator however, the lines begin to get fuzzy. Naturally enough, as we approach the poles, the lines disappear altogether. Thus, we can see the basic features of the standard Mercator projection which has been used for years to create maps of the entire world for navigation and general geographical use.

This is the basic concept of what are called cylindrical projections. This technique takes advantage of the fact that we can take a flat piece of photographic film and make a cylinder out of it, take the picture, and then flatten the film or its image out again. The important concept here is that a cylinder, while round in one dimension, can be easily converted to a perfectly flat surface. A surface which has this characteristic is said to be a developable surface. The cylinder was especially useful for our mapping project here, since it is not only round, but is also developable.

Conic Map Projection

Another surface which has the same characteristic as our previous cylinder is the cone. The cone is round horizontally, but since it is flat vertically, it is also developable. If we just cut it along a vertical line, and unroll it, it will be perfectly flat. Therefore, one might correctly assume that a cone is also a useful surface for projecting the round earth onto a flat map. To visualize this, we'll take some photographic film and make a cone out of it. We'll then place it on the earth as if it were a party hat, as shown in Figure 13-9:

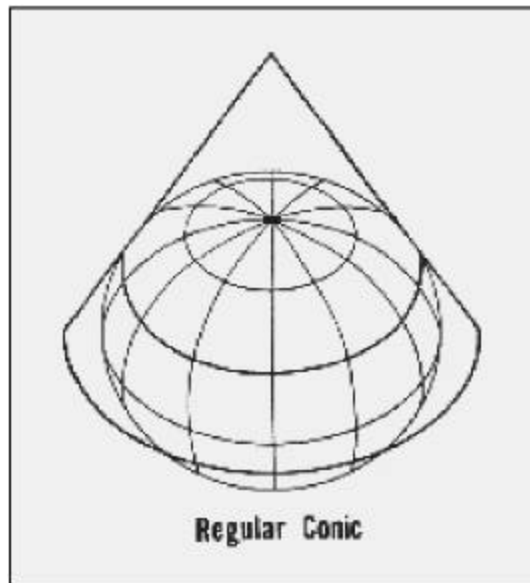


Figure 13-9 Regular Conic Projection

Now, if we expose the film, our photograph will look something like that shown in Figure 13-10:

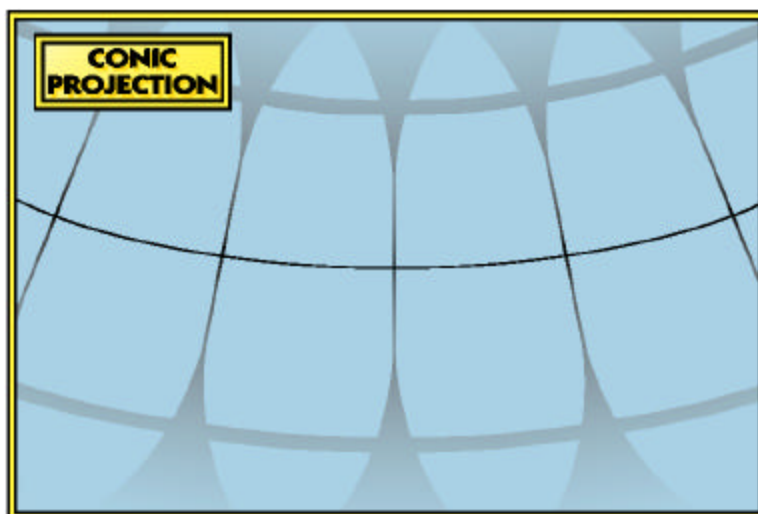


Figure 13-10 Conic Projection Photograph

Notice that the clean crisp lines now follow a certain line of latitude around the earth; the parallel of latitude at which the photographic cone actually touched the surface of the earth. As we proceed either north or south from this parallel of latitude, the lines start to get fuzzy and soon disappear altogether.

We have now briefly discussed the three basic forms of map projections: the Azimuthal, the Cylindrical (Mercator), and the Conic. These three basic forms represent the techniques, or projections, used in almost all of today's mapping. However, there are some interesting variations of these basic forms which we need to examine before we go on.

Transverse Cylindrical Projection

In producing our cylindrical image, we wrapped the cylinder around the earth so that the film touched the earth at the equator and the open ends of the cylinder were toward the north and south poles. Since the earth is a sphere (or nearly so), there are other ways in which our photographic cylinder can be wrapped around the earth. We can also rotate our cylinder 90 degrees, so that the open ends are to the left and right, as shown in Figure 13-11:

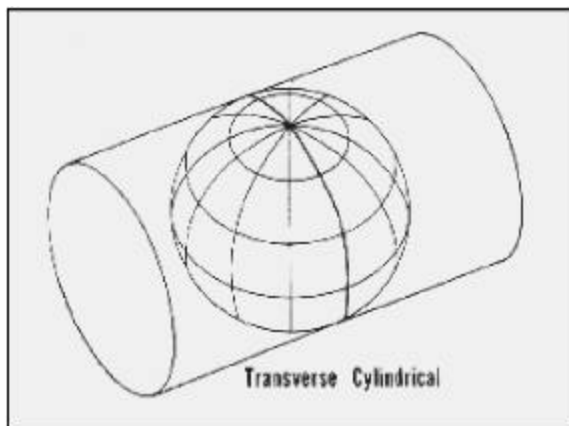


Figure 13-11 Transverse Cylindrical Projection

Now, when we expose our film, we obtain an image as shown in Figure 13:

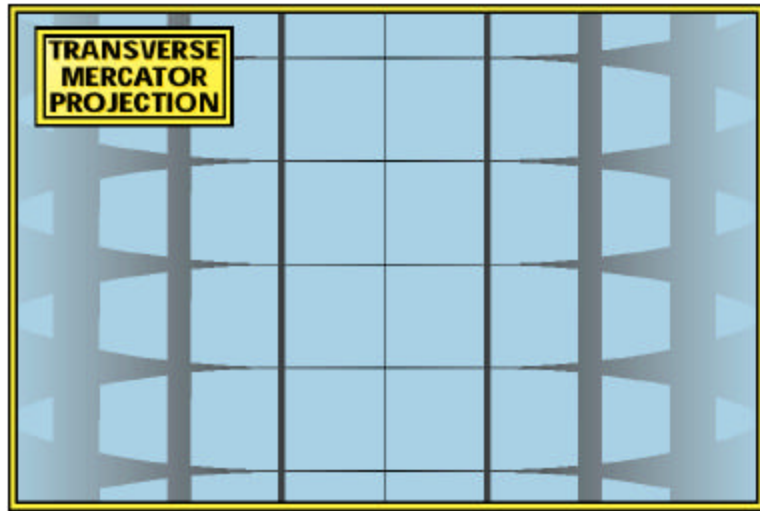


Figure 13-12 Transverse Cylindrical Projection Photograph

The crisp clean lines still represent the regions where the photographic film was close to the earth. But now, instead of an east-west direction, the lines follow a great circle of longitude in a north-south direction. Thus, there is little distortion as we proceed north and south close to the center, but distortion increases as we move to the east and to the west. This is the basic concept of a Transverse Cylindrical projection. This projection is more commonly known as the Transverse Mercator projection in the USA and as the Gauss-Kruger or Gauss Conformal projection in other parts of the world.

The point of all of this discussion is that we have several different cases of the direction and extent of the clean crisp lines on our photographic images. By using different projection techniques, we have achieved different distributions of the projection distortion across our map. The standard cylindrical (Mercator) gives us minimum distortion around the equator. Thus, if one is mapping the temperate zones of the entire earth, it represents a good choice. The Conic enables us to move the east-west line of minimum distortion from the equator to any other line of latitude. Choosing a cone that will touch the earth at about 36 degrees of north latitude would be a good choice for producing a map of the state of Tennessee which is long in the east-west direction. The Transverse Cylindrical (Transverse Mercator) produces clean crisp lines in a north-south direction, thus being the projection of choice for mapping states such as New Hampshire or Idaho that are longer in a north-south direction. Choosing the flat exposure surface, i.e., the Azimuthal projection, provides us with minimum distortion in all directions for a limited distance and this is the projection that is primarily used for maps of the polar regions.

Therefore, minimizing the projection distortion in a map of a specific region of the earth is, to a large degree, a matter of choosing the right projection. The general rule of thumb is this: If the greatest extent of the region to be mapped is east-west, the Conic projection should be used. If the greatest extent of the region is north-south, the Transverse Mercator projection should be used. The brighter folks among us may have conceived the idea that the cylinder (or the cone) could also be tilted, as shown in Figure 13-13 below, to obtain clean crisp lines (i.e., minimum distortion) in any specific direction.

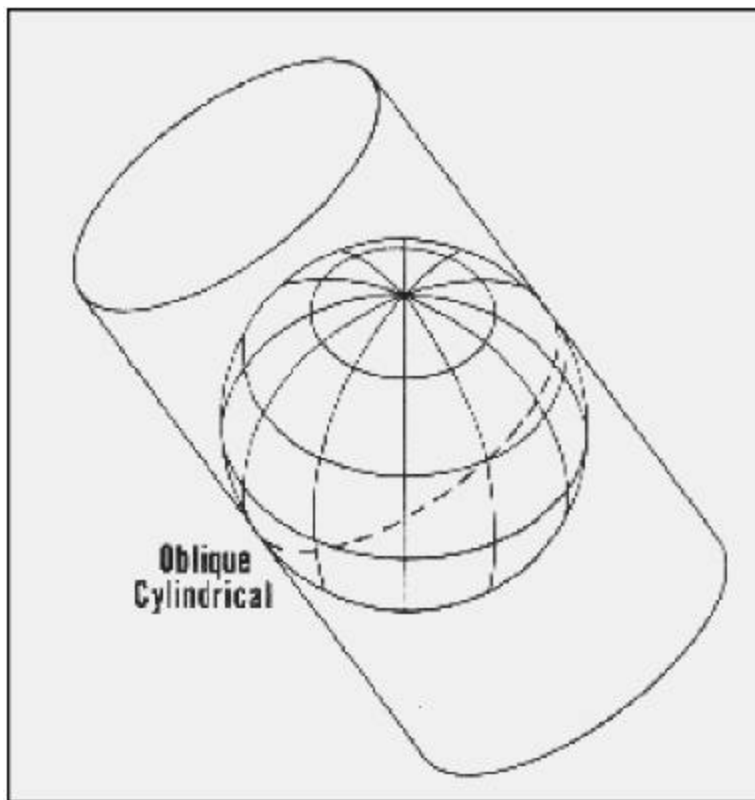


Figure 13-13 Oblique Cylindrical Projection

Such projections do, in fact, exist for the cylindrical case and are referred to as oblique cylindrical projections. For example, the Hotine Oblique Mercator projection is used to map areas that do not run either north-south or east-west, such as the Alaskan panhandle. Since the Hotine projection is used only occasionally, we'll discuss it only briefly, and later on in this summary.

We have seen that the degree of projection distortion at any given point on a map depends upon how far away the developable surface is from the surface to be mapped. Furthermore, we have seen that we can get the photographic film closer to the surface being mapped by using different shapes. There is also another technique used in cartography to reduce the overall distance between the photographic film and the surface of the earth for a given region. In the case of a cylindrical projection, imagine the result if we could shrink the cylinder into the earth just a little bit. Some of the cylinder would be inside the earth and some would be outside of the earth. However, for a specific region of the earth, the largest distance between the photographic film and the globe will be reduced. Further, there would be two lines at which the cylinder actually touched the surface, thereby giving us two lines of actual zero distortion, instead of just one. This little technique enables us to, in general, get more of the film closer to more of the region that we want to map.

In the case of the cylindrical projection, this shrinkage is usually specified as the Scale Reduction Factor. It is also referred to, for reasons which will become clear later on, as the Scale of the Central Meridian. These values are usually very close to one; typically in the range of 0.9996 to 0.9999, depending upon the size of the region being mapped. Figure 13-14, below, shows the result of the scale reduction when it is applied to a Transverse Mercator projection of the USA:

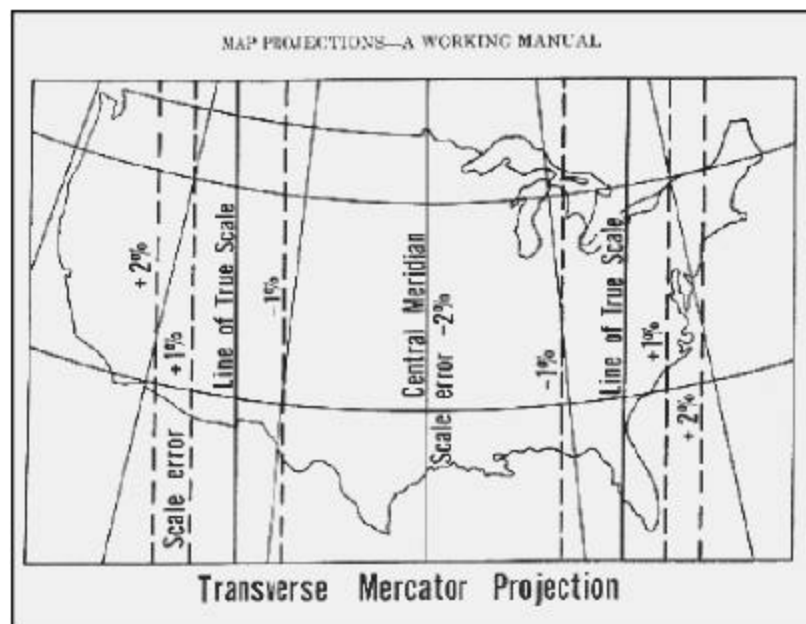


Figure 13-14 Transverse Mercator Projection Showing A Scale Reduction Factor of 0.9800

Note the two north-south lines of true scale, where there is no distortion when using this projection system.

A similar technique is used for conic projections. However, instead of specifying a shrinkage factor, the shrinkage is usually specified by giving the latitude at which the cone enters the earth and the latitude at which the cone leaves the earth. These latitudes are referred to as the Standard Parallels of the projection, or the Northern and Southern Parallels of the projection. The effect is the same as that achieved in the cylindrical case. The largest distance between the projection surface and the region being mapped is substantially reduced, and again, two lines of zero distortion are obtained. The affect of shrinkage in conic projections is demonstrated in the depiction of the Lambert Conformal Conic Projection in Figure 13-15:

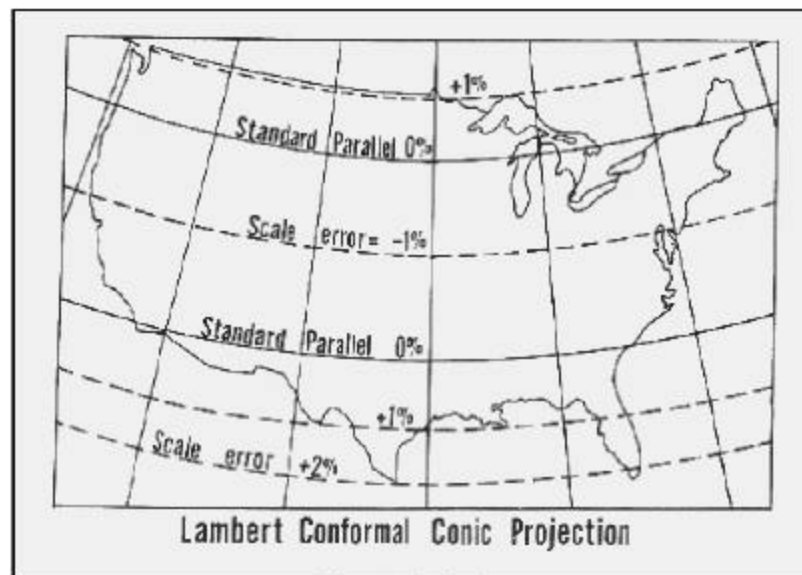


Figure 13-15 Lambert Conformal Conic Projection With Two Standard Parallels

The Lambert Conformal Conic Projection (often called the Lambert Grid when using state plane coordinates) was developed in 1772 by Johann Heinrich Lambert, but was rarely used until World War I, when the French army adopted it for their battle maps. However, today it is used for a majority of the state plane coordinate systems in the U.S. and in other countries or regions that are predominantly east-west in expanse. We will discuss it in more detail, later.

We have now explored several different techniques of projecting the round earth onto a flat piece of photographic film. If were to go to work as cartographers, we would perform essentially the same process. Of course, we wouldn't use photographic film or put a

strobe light at the center of the earth. We would simply emulate this entire process by using mathematics. The projections we have just discussed are used for 95% of mapping worldwide and are shown in Figure 17 for your review:

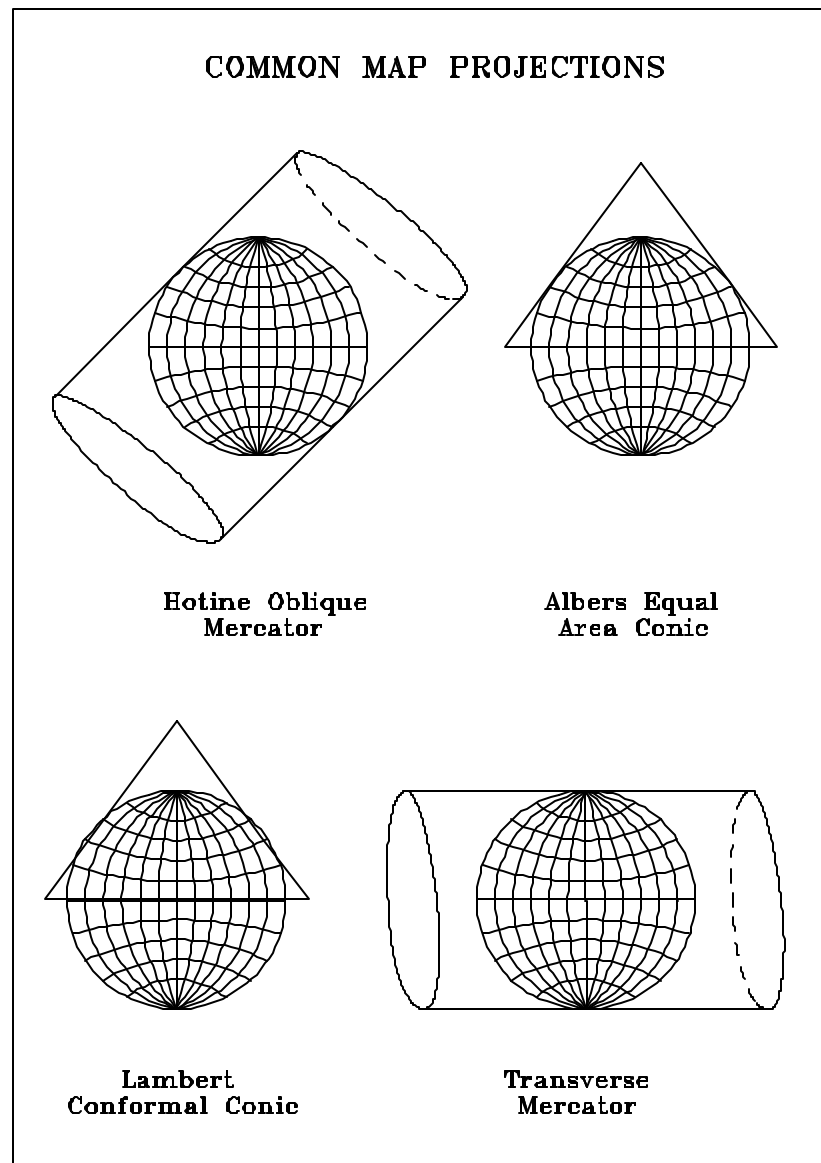


Figure 13-16 Most Common Map Projections

To summarize, a cartographic (map) projection is the mathematical process of projecting the spherical geographic coordinates of latitude and longitude onto a developable surface and then flattening it out. Since mathematics, and also computers, deal largely with numerical quantities, we must establish what types of numbers we will be converting these values to. Since **Terramodel** works in cartesian coordinate systems, we will want to produce cartesian coordinates, i.e., the X and Y coordinates that we have been taught to use since junior high school. These are very convenient, since we all know how to compute the length of a line, given its cartesian coordinates. Similarly, we know how to compute areas, perform various coordinate geometry functions, and, in general, manipulate cartesian X and Y (or easting and northing) data to produce our engineering design drawings and land survey plats.

For the mathematics that we must use to convert the spherical latitude and longitude coordinates to X and Y linear cartesian coordinates, we need to know the radius of the earth. As we have discussed previously, with all the electronics and satellite data (GPS) available today, we know what the true radius of the earth is. However, as we also noted, this has not always been so. Imagine you were a geodesist in the early part of this century and you were asked to measure the size and shape (i.e., the exact radius) of the earth. How would you proceed? The methods used were based on a variety of techniques and involved exhaustive painstaking measurements, usually astronomical in nature. There were many determinations of the radius of the earth; all of which were very close but none were the same. Regardless, you must have a value of the radius of the earth if you want to convert an angular measure of, for example, a specific latitude/longitude coordinate to a linear measure, such as cartesian coordinates in meters or feet.

As noted previously, two of these measurements are in general use in the United States today. The measurements made by Clarke in 1866 are used for all maps and digital data produced before the advent of the North American Datum of 1983, and a lot of maps produced since then. Therefore, any geodetic data that isn't very new is probably based on the Clarke 1866 ellipsoid. Since Clarke also made measurements in 1880, the year is also given to distinguish which measurement is to be used. Recently, through the use of satellite data and sophisticated electronic measuring techniques, a very accurate determination of the size and shape of the earth has been made. This is referred to as the GRS 1980 ellipsoid; i.e., Geodetic Reference System of 1980.

In converting from latitude and longitude to a linear unit of measure, we are converting from angular units (usually degrees) to a linear unit such as meters or feet. The units we use for the radius will determine the unit of linear measure. Therefore, since the size of the ellipsoid is usually given in meters, we must convert the radius from meters to feet if we want our cartesian X's and Y's to be in units of feet. As most land surveyors are aware, there are two different units of feet used in cartography. The current official definition of a foot is based on the definition that one inch equals 2.54 centimeters exactly. This definition is usually referred to as the International Survey Foot, and is rarely used in cartography in the United States. Rather, the foot most often used is the U.S. Survey

Foot which is based on the older definition of one meter equals 39.37 inches exactly. The difference is small, but when dealing with the large numbers used in cartography, the difference can be very significant.

So far, we have described three choices which must be made to rigorously define a process by which geographic coordinates can be converted to cartesian coordinates. These are: (1) A choice of the projection method, (2) The degree of shrinkage or the scale factor, and (3) A selection as to the earth's ellipsoid which is to be used. We have some additional choices to make before we are finished, but first let's look at a vehicle that we can use to package all of these choices together.

The term cartographic coordinate system refers to a collection of choices that determine exactly how a set of geographic coordinates (such as latitude and longitude) are, have been, or will be converted to cartesian coordinates. The definition includes all information necessary to complete the process, and invert or reverse it when necessary.

As noted previously, we have determined that a coordinate system definition must include a choice of the projection method, the scale reduction factor, and the type of earth's ellipsoid. The remaining choices you must make depend upon the specific projection that you selected. We will discuss the two most common projections, the Transverse Mercator, and the Conic. These two projections account for a majority of the cartographic work done in the United States.

The Transverse Mercator Projection

Let's start our discussion of the Transverse Mercator projection system by setting up a coordinate system definition for use in mapping the state of New Hampshire (since the north-south extent of New Hampshire is significantly larger than the east-west extent). In selecting an ellipsoid, we notice that the geographic data that we are converting is based on the North American Datum of 1927, so we choose the ellipsoid upon which all of that information is based, the Clarke ellipsoid of 1866. Recall that the Transverse Mercator projection is based on a cylinder wrapped around the earth so that the cylinder touches the earth on a single great circle of longitude, a single meridian. To achieve minimum distortion, it is customary to wrap the cylinder around the meridian which is in the center of the region to be mapped. Thus, we refer to this meridian as the Central Meridian. Our coordinate system definition must include, therefore, a specification of the Central Meridian, i.e., the longitude value which is in the center of the region to be mapped. In our example of New Hampshire, the meridian at $71^{\circ} 40'$ west longitude is chosen, since it approximately bisects the state.

Since the shrinkage concept we talked about earlier is almost always used when mapping a large area in order to minimize distortion, we must consider the amount of shrinkage, or Scale Reduction, that is appropriate for New Hampshire. Analytically, we want a Scale

Reduction Factor which produces, overall, the smallest distance between the projection surface and any point in New Hampshire, the region to be mapped. To avoid any heavy mathematics, we choose a Scale Reduction Factor which will give us our two lines of zero distortion about 1/4 of the map in from the west and 1/4 of the map in from the east. Since New Hampshire is close to 2 degrees wide at its widest point, we want our Scale Reduction Factor to produce a cylinder which intersects the earth about 1/2 of a degree on either side of the Central Meridian. Therefore, we set our shrinkage factor to approximate the cosine of 1/2 of a degree, or 0.99996. The shrinkage is most often expressed as a ratio, such as 1 part in 30,000 for our case of New Hampshire.

Our decision with regard to the Central Meridian defined the longitude value at which the projection mathematics will produce a zero X coordinate. We also must determine which latitude should produce a Y coordinate of zero. This selection is quite arbitrary, but since it is convenient to work with numbers that are always positive, a latitude outside of, and to the south of the region to be mapped is customarily selected. For New Hampshire, 42° 30' north latitude is south and outside the state and represents a good choice and a nice round number.

The definition of our coordinate system for New Hampshire is almost complete except for one item of convenience. Since our central meridian bisects the state, about half of our X coordinate values will be negative. Cartographers consider this to be an inconvenience (as do many plotting devices), so coordinate system definitions usually include a false origin or an offset which is added to all coordinates to cause all of them to be positive. The X component of this offset or false origin is referred to as the False Easting, and the Y component is called the False Northing. The choice is somewhat arbitrary and depends upon the size of the region to be mapped. Since the choice is arbitrary, a round number which is guaranteed to adjust all X numbers into the positive range is customarily chosen. For our map of New Hampshire, a False Easting of 500,000 will achieve the desired result. This meets our requirement of causing all X coordinates to be positive and is a nice round number which is easy to communicate and work with. Since we chose our origin latitude to be south of the New Hampshire border, all of the Y coordinates are guaranteed to be positive already, so a False Northing of zero is suitable.

Finally, we need to set the scale of our coordinate system. Paper maps require that any area to be mapped be scaled down so that it fits on the piece of paper we intend to use. **Terramodel** does not have a paper size problem, since it can easily emulate an infinitely large piece of paper. Since **Terramodel** has no problem with units and we would like to have the benefit of working in real world units, we will set the scale of our coordinate system to one. With a scale of one, the length and area calculations will produce real-world results in the same units used to specify the radius of the earth.

This completes the definition of our coordinate system which is based on the Transverse Mercator Projection. Let's take a moment to review the requirements of this definition. These requirements specify what the mathematics of the projection will need to know in order to convert geographic coordinates to the cartesian X's and Y's.

Equally important, the mathematics of the projection will need to know these items if the cartesian data in **Terramodel** is to be converted back to geographic coordinates.

The specification of our coordinate system for New Hampshire that is based on the Transverse Mercator Projection is shown below:

Table 13-4 Transverse Mercator Projection Parameters

Coordinate System Parameter	Example Value, New Hampshire State Planes
Projection	Transverse Mercator
Ellipsoid	Clarke 1866
Linear Unit	U. S. Survey Foot
Central Meridian	71° 40' West Longitude
Scale Reduction Factor, or Scale of the Central Meridian	1:30,000 or 0.99996666667
Origin Latitude	42° 30' North Latitude
False Easting	500,000
False Northing	0
Scale	1:1 or 1.0

Table 13-4 shows all of the elements required to define a Transverse Mercator Projection. If you have surveyed in New Hampshire, you will probably recognize that the coordinate system we have just developed is that of New Hampshire State Plane Coordinates based on the North American Datum of 1927.

A Brief Summary of the Transverse Mercator Projection

The Transverse Mercator is a variation of the Mercator projection. In the Mercator projection, a cylinder is wrapped around the earth tangent to the sphere along the plane of the equator. The projection is cylindrical and conformal. However, in the case of the Transverse Mercator, the cylinder is rotated 90 degrees so that it is tangent to the sphere at a chosen meridian. In general, the Transverse Mercator is used in state plane zones in those states having a greater north-south areal extent than east-west. It is also the basis of the Universal Transverse Mercator coordinate systems.

When the Transverse Mercator projection is displayed, the central meridian is a straight line, as are all meridians precisely 180 degrees away from the central meridian of a projection. All other meridians and parallels are considered to be complex curves.

The scale is accurate along the central meridian if no scale reduction factor is used. A scale reduction factor recesses the cylinder slightly into the ellipsoid. Two lines of true scale, parallel and equidistant to the central meridian are produced. The scale reduction factor evenly distributes scale distortion over a zone.

This projection requires coordinates for a central meridian, a scale reduction factor (which can be set to 1.0 if scale reduction is ignored), an origin latitude, and a false easting and a false northing. See Figure 13-17 for a visual review of the areas found in a Transverse Mercator State Plane Coordinate System:

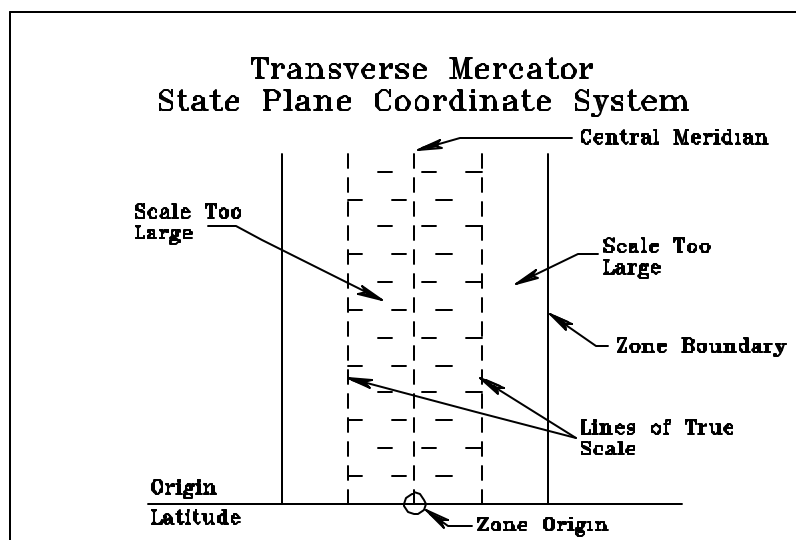


Figure 13-17 Transverse Mercator System

Now, let's look at a further variation of the Mercator projection: the Universal Transverse Mercator.

The Universal Transverse Mercator Projection

UTM is an acronym for Universal Transverse Mercator- a series of 120 coordinate systems established by the US Army in 1947 for the purpose of constructing world-wide military maps between 80 degrees south and 84 degrees north. North and south of this zone, Universal Polar Stereographic coordinate systems (not supported by **Terramodel**) are used.

UTM zones do not exist north and south of about 80 degrees because map direction becomes ambiguous. Up is no longer north, and right is no longer east. At the poles, all northern (or southern) UTM central meridians intersect, and zone designations have no meaning.

As the name implies, the UTM system is based on the Transverse Mercator projection. Each zone is six degrees wide and is numbered. Zone 1 covers 180° to 174° west longitude with a central meridian at 177° west. Zone 2 covers the region between 174° and 168° west and has a Central Meridian at 171° west longitude. In similar manner, the entire northern hemisphere is covered by a system of 60 zones, marked off consecutively from west to east. The scale reduction factor for each zone is 0.9996, the latitude of origin is the equator, the linear unit is the meter, and the False Easting is 500,000 meters. For the northern hemisphere, the False Northing is zero. A similar set of 60 zones exists for the southern hemisphere. The only difference for the southern hemisphere zones is that the False Northing is set to 10,000,000 in order to obtain positive Y coordinate results.

A purposeful scale factor of 0.9996 is applied to UTM coordinates. This has the effect of distorting the map along the central meridian by a small amount, but also creates two lines of intersection having true scale 1.5 degrees from the edge of each zone. Mathematically, this has the effect of distributing the amount of distortion over the zone instead of accumulating it along the edges of the zone.

The reference ellipsoid used for UTM zones varies from country to country and in a single state. In the conterminous United State and in Alaska, the Clarke 1866 ellipsoid is used to determine UTM coordinates. For the UTM grid superimposed on the state of Hawaii, however, the International Ellipsoid is used.

To define a UTM zone, the location of the central meridian, and the scale reduction factor (always 0.9996) must be known. The origin of each UTM zone is always located at the intersection of its central meridian and the equator.

Since being defined by the US Army, these coordinate systems have become quite popular and you will often see data being made available that uses this set of coordinate systems. The projection distortion of these coordinate systems is roughly less than 1 part in 3,000. See Figure 13-18 for the areas defined in a Universal Transverse Mercator Coordinate System:

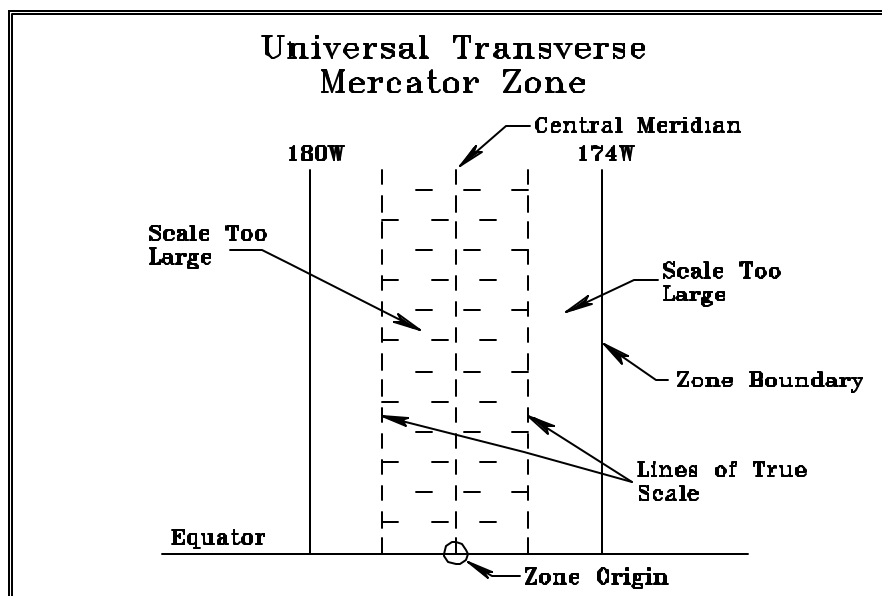


Figure 13-18 Universal Transverse Mercator Coordinate System Areas

A Brief Summary of the Universal Transverse Mercator Projection

Now let's briefly summarize the characteristics of both the Transverse Mercator projection and the Universal Transverse Mercator projections. For both projections:

- are transverse cylindrical projections.
- are conformal (both accurately preserve the shape of mapped entities within a small area).
- have a linear scale that is true along the line of tangency or along two lines equidistant from and parallel to the line of tangency.
- have a central meridian, and it, as well as each meridian 90 degrees from the central meridian, and the Equator are all straight lines.

- All other meridians and parallels are complex curves.
- They are used extensively for quadrangle maps at scales of from 1:24000 to 1:250000.
- They are used where the north-south dimension is greater than east-west dimension.

While the Transverse Mercator is the projection of choice for most states that are predominantly north-south, those states that run primarily east-west require a different treatment. The projection chosen for those states is usually the Lambert Conformal Conic Projection.

The Lambert Conformal Conic Projection

Tennessee is different from the previous state for which we established a projection system (New Hampshire) in that its east-west extent is significantly larger than its north-south extent. Therefore, we would most likely choose a conic projection (such as the Lambert Conformal) for that state. We also need to choose an ellipsoid and linear unit as before. In conic projections, we specify the angle of the cone and the shrinkage, along with a set of Standard Parallels, Northern and Southern. Cartographers generally set the parallels one sixth of the map's extent in from the north and south edges of the region to be mapped, while choosing round numbers. For our map of Tennessee, if we use these guidelines, values of $36^{\circ} 25'$ and $35^{\circ} 15'$ of north latitude are reasonable values for the Standard Parallels.

The longitude origin of the conic projection specifies the one line of longitude which will be straight and perfectly vertical. The projection mathematics will also convert this longitude value to an X coordinate of zero. This is usually chosen to be the center of the map, in round numbers. For Tennessee, 86° west longitude is a reasonable number. As with the cylindrical projection that we just completed for New Hampshire, the latitude of the projection origin is often chosen to be south of the southernmost extent of the region to be mapped. A round number is usually used, so $34^{\circ} 40'$ is a reasonable value for Tennessee.

False Eastings and False Northings are again chosen so as to force all coordinates to be positive. Since the Conic projection is usually used to map regions whose east/west extent is large, the False Easting is usually much larger than that encountered in coordinate systems based on the Transverse Mercator Projection. For Tennessee, a value of 2,000,000 for false easting is reasonable, to avoid negative eastings. Now, just to demonstrate the arbitrary nature of False Eastings and Northings, we'll set our False Northing for Tennessee at 100,000. The following table shows the definition of a state coordinate system for Tennessee that uses the Lambert Conformal Conic Projection:

Table 13-5 Lambert Conformal Projection Parameters

Coordinate System Parameter	Example Value, Tennessee State Planes
Projection	Lambert Conformal Conic
Ellipsoid	Clarke 1866
Linear Unit	U. S. Survey Foot
Northern Parallel	36° 25' North Latitude
Southern Parallel	35° 15' North Latitude
Origin Longitude	86° 00' West Longitude
Origin Latitude	34° 40' North Latitude
False Easting	2,000,000
False Northing	100,000
Scale	1:1 or 1.0

This definition includes all of the necessary components of a conic projection and, by intent, exactly defines the State Plane Coordinate System for the State of Tennessee.

A Brief Summary of the Lambert Conformal Conic Projection

The Lambert Conformal Conic Projection has the following properties:

- Its projection is based on a cone that is conceptually secant on two parallels.
- It is conformal in nature (areal distortion is minimal but increases away from the standard parallels).
- Its parallels of latitude are unequally spaced, and are closer together at the center of the map.
- Its meridians are equally spaced and intersect the parallels at right angles; its meridians are straight lines, converging at a pole.

- The scale of the map is true along the two ordinarily chosen standard parallels.
- It is used for maps of states and countries (such as the U.S.) that run predominantly east-west in extent.
- While Johann Heinrich Lambert published this projection in 1772, it was overlooked by the Coast and Geodetic Survey until the 1930's.

Applications for the Lambert Conformal Conic Projection include 1:1,000,000 world aeronautical charts, 1:500,000 state base maps, and 1:24,000 7.5 minute quadrangle maps in Lambert states of east-west extent (after 1957). This projection is also used extraterrestrially to map the middle latitudes of the Moon, Mercury, Mars, and Jupiter's satellites.

The Lambert Conformal Conic Projection requires the location of two reference parallels, an origin latitude, and origin longitude, a false easting, and a false northing. See Figure 13-19 for a visual summary of areas defined by a Lambert State Plane Coordinate System:

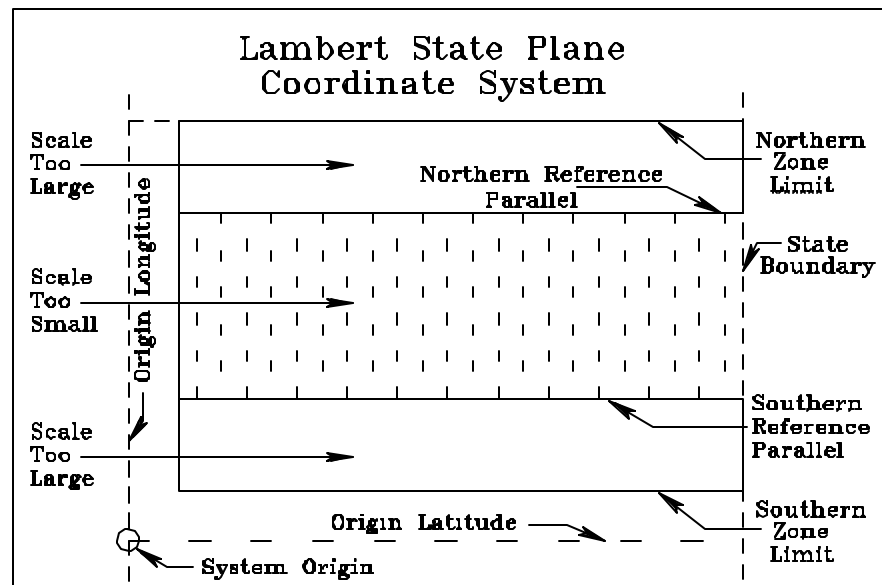


Figure 13-19 Lambert State Plane Coordinate System Areas

The Hotine Oblique Mercator Projection

The Hotine Oblique Mercator Projection is an oblique conformal projection. This projection is thought to be an adaptation of the Mercator and Transverse Mercator and is used for localized areas that are oriented other than north-south or east-west (such as the Alaskan panhandle). The British geodesist Martin Hotine, published a form of this projection in 1946-47. This projection is used as the basis of one Alaskan state plane zone, so we will discuss it very briefly.

To use the Hotine Oblique Mercator projection, the location of a great circle other than a meridian or the equator must be defined. (A great circle is one whose center lies at the center of the earth.) The 45th parallel, for example, is not a great circle but the equator is, just as all meridians of longitude are great circles.

There are two different ways to specify the required great circle. The first method requires the latitude and longitude coordinates of a single point on the ellipsoid and an azimuth (degrees east of north) of a great circle from the point. The second method requires two points on the ellipsoid that define a great circle. It can be shown that between any two points on a sphere, a great circle exists to connect them.

One of the restrictions placed on the Hotine Oblique Mercator projection can be described as follows. Imagine that you are traversing a great circle and noting your latitude every so often. At some point, your latitude stops increasing (or decreasing) and starts decreasing (or increasing). The two points cannot straddle the line where the latitude stops increasing and starts decreasing.

One more caveat is in order. When traversing the great circle and noting the latitude every so often, and the direction of change in latitude reverses before reaching the equator, you are proceeding towards the wrong one of the two possible origins. Go the other way.

We won't discuss it here, but you should be aware that there are two variations of Hotine formulas that can be used to produce two distinctly different types of coordinates.

Use of the Hotine Oblique Mercator projection requires specification of the latitude and longitude of a single point and an azimuth or two points on a sphere. Since it is a variation of the Transverse Mercator projection, you can view Figure 13-14 for a representation of its elements. Just remember that the Hotine Oblique Mercator projection does not follow a north-south central meridian, but instead follows a great circle at a skewed angle.

A Brief Summary of the Hotine Oblique Mercator Projection

To summarize the Hotine Oblique Mercator projection, it has the following properties:

- It is cylindrical, but oblique, i.e. not running due north-south or east-west.
- It is conformal (relative local angles are correct).
- Shape is true only within any small area.
- Only two of its meridians, 180 degrees apart are straight lines.
- It other meridians and parallels are complex curves.
- Its scale on the spherical form is true along a chosen central line, a great circle at an oblique angle, or along two straight lines parallel to the central line. The scale on the ellipsoidal form is similar but varies slightly from this pattern.
- It has been used for mapping the Alaskan panhandle, the five Great Lakes, the St. Lawrence River, an area along the U.S.-Canada border east of Lake of the Woods. More recently, it has been used for mapping Landsat, weather satellite imagery, and the National Geographic Society's maps.
- It was developed from 1900 to 1950 by Rosenmund, Laborde, Hotine, and others.

State Plane Coordinate Systems

We've been discussing them theoretically, but just what are state plane coordinate systems? We have technically defined two state plane coordinate systems here, but we defined them with a broader definition, also. Essentially, state plane coordinate systems are coordinate systems, established by legislation, and in common use by land surveyors, civil engineers, planners, plus state and local governments. They provide a common basis for assigning coordinate values to all of the areas in each of the respective states and a maximum allowable projection distortion is maintained for each system in the state.

State plane coordinate systems are generally required to be defined so as to maintain a projection distortion of less than one part in 10,000. As a result, several of the larger states need several zones (Alaska has 10, for example), in order to meet this criteria. With only one exception (the Alaskan panhandle), all state plane zones are based on either the Lambert Conformal Conic Projection or the Transverse Mercator Projection. In the jargon of state plane coordinate systems, Colorado is a Lambert state with three zones: a south zone, a central zone, and a north zone. Wyoming is a Mercator state with four zones: an east zone, an east central zone, a west central zone, and a west zone. California is different from most states. While its extent is primarily north-south, it has seven zones (six based on NAD83) all based on the Lambert Conformal Conic (instead of Transverse Mercator) and the zones are numbered, usually with Roman numerals.

To recap our discussion, a cartographic process is necessary to provide **Terramodel** with cartesian data which will provide data for the analysis, interpretation, and decision-making processes for which we use these systems. Choosing the appropriate coordinate system for your cartographic process, and the implied projection selection, is an important part of maintaining the accuracy and usefulness of the data in your **Terramodel** database. **Terramodel** supports all of the appropriate officially-defined State Plane Coordinate and northern hemisphere UTM systems in common use today and will allow you to easily convert data from Cartesian coordinates to latitude/longitude and vice versa. Use great care in moving coordinates from one coordinate system to another. While this can usually be done fairly accurately between two coordinate systems in the same state, when moving from one state to another or from one ellipsoid to another, you may find variations. Also, when acquiring data for use in **Terramodel**, be sure you know what coordinate system was used to generate the X and Y (or easting and northing) coordinates of the data.

Most of the information presented here, and many of the graphics, were extracted from a paper titled "Choosing a Coordinate System for Your GIS/CAD System" by Norman T. Olsen of Mentor Software, Inc. (Mentor Software, Inc. is the developer of all of the underlying algorithms in **Terramodel** that are used by the **Coordcon** command to convert coordinate data.) Norman Olsen's paper and some graphics, in turn, were based

in part on "Map Projections, A Working Manual" by John P. Snyder. Snyder's work is also known as U. S. Geological Survey Professional Paper 1395. Certain excerpts have also been taken from Mentor Software's "Tralaine User's Manual". Finally, some of the supplementary background material came from "The Surveyor's Handbook" by Russell C. Brinker and Roy Minnick.

How to Convert Coordinates Using Coordcon

Before Using CoordCon

Coordcon simplifies the process of converting coordinates, but there are guidelines you should follow.

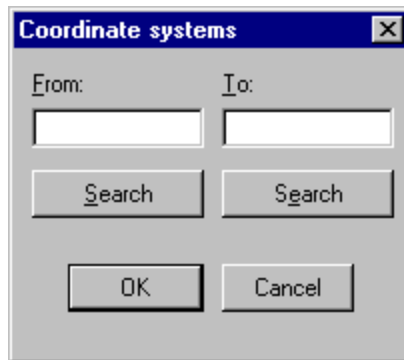
Before performing a coordinate conversion, be sure to make note of the following:

- Set the **Feet to meters** factor in the **UnitsSet** command (UnitsSet/Conv. factors/Meters/foot) before converting or listing coordinates. **Coordcon** uses this setting when converting from the US Survey foot or the International Survey foot to meters, and vice/versa. The conversion factor for converting from U.S. Survey feet to meters is .3048006096. The conversion factor for converting from International Survey feet to meters is .3048.
- When converting between coordinate systems based on NAD-27 and NAD-83, **Terramodel** is accurate to within about 15 centimeters (RMS) in the contiguous 48 states since it uses the NADCON overlay for the conversion. The NADCON overlay includes data for the conterminous 48 states as well as Puerto Rico and the Virgin Islands. You can obtain data files for Alaska or other areas up to 200 miles offshore of the 48 conterminous United States by requesting the NADCON Overlay Diskette directly from the National Geodetic Survey. Copy this diskette into the **Terramodel** directory. The file containing the latitude datum shift will have a suffix of .LAS, while the longitude datum shift file will have a suffix of .LOS. These are also available from the internet at <http://www.ngs.noaa.gov> or <ftp://ftp.ngs.noaa.gov/pub/pcsoft/nadcon> and should be updated as needed.
- Always make a backup copy of your project file prior to converting coordinates for the points in your file.
- Use the **FactZ** command to scale the elevations of objects from one set of units to another. Use the **Scale** command to scale the X,Y coordinates from one set of units to another.
- When converting point objects from Lat Long to other systems, the X and Y coordinates of the points must be in decimal degrees. A point with a coordinate of 30 degree 30 minutes would have a corresponding coordinate of 30.5.

- We recommend that you perform any transformation on ALL of the points in your file. Converting only some of the points may result in mixed coordinates in the project file and cause errors in design.

To convert coordinates:

1. Type CoordCon into the command bar.
OR
Click COGO, Utilities, and Coordinate Conversions.
The *CoordCon* command bar appears.
2. Click Coordinate Systems. The *Coordinate Systems* dialog box appears.



3. Type the name of the coordinate system on which your point(s) is (are) based in the From box.
OR
Click on Search to select a coordinate system.
4. Type the name of the coordinate system to which you want your point(s) converted.
OR
Click Search to select a coordinate system.
5. Click OK. Terramodel is now set up for the conversion.
6. To see the coordinate conversion listing for a single point:
 - a. Click Any Point. The command bar changes.

-
- b. Use the Loc point coordinate control to input the point. The location appears in the Loc box.
 - c. Click List. The Point Conversion Listing dialog box appears with the desired information.
 - d. Click OK to close the dialog box.
 - e. Repeat steps b-d for any other point. Click Close when you are done viewing one point at a time.
 7. To see the coordinate conversion listing for multiple points:
 - a. Click Multiple Points. The command bar changes.
 - b. Use the Select Points select control to select the points.
 - c. Click List. The P3Pad viewer appears with the desired information.
 - d. Click FILE and Close to close the viewer. For help while using the viewer, press F1 while the view is open.
 - e. Repeat steps b-d for other points. Click Close when you are done viewing multiple points.
 8. To convert one or more points from one coordinate system to another:
 - a. Click Multiple Points. The command bar changes.
 - b. Use the Select Points select control to select the points for coordinate conversion.
 - c. Click Convert. A dialog box appears telling you the conversion is complete. Click OK.
 - d. Repeat steps a-c for other points. Click Close when you are done converting points.
 9. Click Close to close the command bar.

Search Coordinate Systems Dialog Box

The Search Coordinate Systems dialog box is used to let you select a coordinate datum and system for conversion. If your coordinate system is not listed, you can define your own system.

To use the Search Coordinate Systems dialog box:

1. Click Search in the *Coordinate Systems* dialog box. The *Search Coordinate Systems* dialog box appears.



2. Click Define CS to define your coordinate system. See *Defining a Coordinate System for Conversion* below.
3. To choose a standard coordinate system available through Terramodel:
 - a. Click the option button that best describes the datum your coordinate system uses. (Click Other if the datum is not listed.)
 - b. Click the Select System box, and type the coordinate system name.
 - c. To list the specifications of the available coordinate systems, click List All.

OR

To confirm you have chosen the correct coordinate system, click List Selected to review its specifications.

4. To choose a coordinate system you have defined:
 - a. Click Define CS. The User Defined Coordinate Systems Dialog box appears.
 - b. Click the Coordinate System Name box. Type in or click the name of the coordinate system.

- c. Click Set Current.
5. Click OK.

Defining a Coordinate System

You can define a coordinate system for use in coordinate conversion.

To define a coordinate system:

1. If you have not opened the *CoordCon* command bar:
Type **CoordCon** into the command bar.
OR
Click COGO, Utilities, and Coordinate Conversions.
The CoordCon command bar appears.
2. Click Coordinate Systems, and click Search. The *Search Coordinate Systems* dialog box appears.



510 How to Convert Coordinates Using Coordcon

3. To see a list of the available ellipsoids or to define your own ellipsoid, click Define Ellipsoid.
4. To see a list of the available datums or to define your own datum, click Define Datum.
5. Click Define CS. The *User Defined Coordinate System* dialog box appears.

User define coordinate system

Coordinate system name: BPCNC

Name of datum: <NONE>

Name of Ellipsoid: BPCNC

Description: Bipolar Oblique Conformal Conic

Origin:

Latitude: 0.0000000000

Longitude: 0.0000000000

False northing: 0.0000000000

False easting: 0.0000000000

Scale reduction: 1.0000000000

Units of system: METER

Projections: <NONE>

Param 1: -110.00000000

Param 2: -20.00000000

Param 3: 0.0000000000

Param 4: 45.0000000000

Buttons: Set Current, New CS, Cancel

6. Click the Name of Datum box, and click the datum you are using to define coordinate system.
OR
Click the Name of Ellipsoid box, and click the ellipsoid you are using to define coordinate system.
7. Click the Coordinate System Name box, and type the name (up to 17 alphanumeric characters) of your coordinate system.
8. Click the Description box, and type an appropriate description, so you can more easily identify the coordinate system later.
9. Click the Projections box, and click the projection you are using to define this coordinate system. The boxes that require information become active, and are relabeled appropriately.

10. Type the required information into the active boxes.
11. Click New CS. A notice box appears telling you the coordinate system was added to the dictionary. Click OK. If this notice box states your coordinate system was not added to the dictionary, some required information was either mistyped or missing.
12. Click Set current to select the coordinate system.

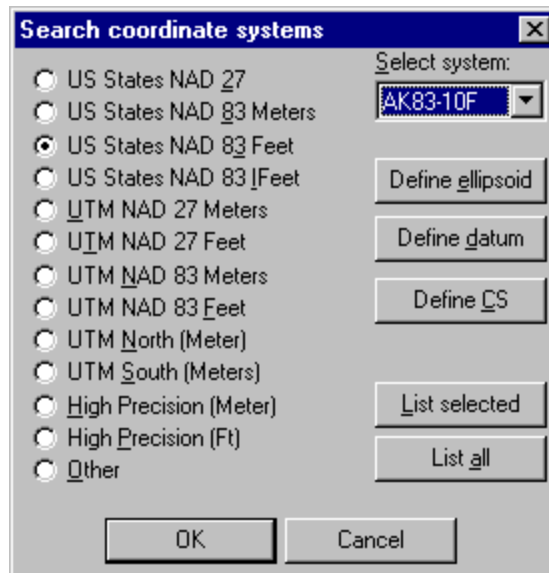
Defining an Ellipsoid

You can see a list of the available ellipsoids with their definitions or define your own ellipsoid using the User Defined Ellipsoid dialog box.

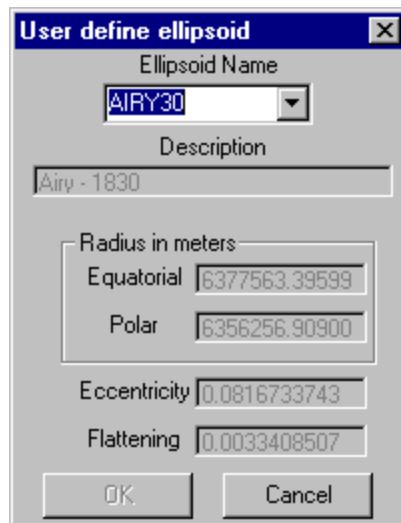
To define or view ellipsoids:

1. If you have not opened the *CoordCon* command bar:
 - Type *CoordCon* into the command bar.
 - OR
 - Click COGO, Utilities, and Coordinate Conversions.
 - The *CoordCon* command bar appears.
2. Click Coordinate Systems, and click Search. The *Search Coordinate Systems* dialog box appears.

512 How to Convert Coordinates Using Coordcon



3. Click Define Ellipsoid. The *User Defined Ellipsoid* dialog box appears.



4. To select an existing ellipsoid, click the Ellipsoid Name box and click the name of the ellipsoid.
5. To define a new ellipsoid:
 - a. Type the name of the ellipsoid you want to define in the Ellipsoid Name box (up to 22 alphanumeric characters).

-
- b. Type a description for the new ellipsoid into the Description box (up to 64 alphanumeric characters, including spaces).
 - c. Type the Equatorial and Polar Radius values into the appropriate boxes.
6. Click OK. CoordCon calculates the Eccentricity and Flattening values from the Equatorial and Polar Radius values. The ellipsoid is ready for you to use when you define your coordinate systems.

Defining a Datum

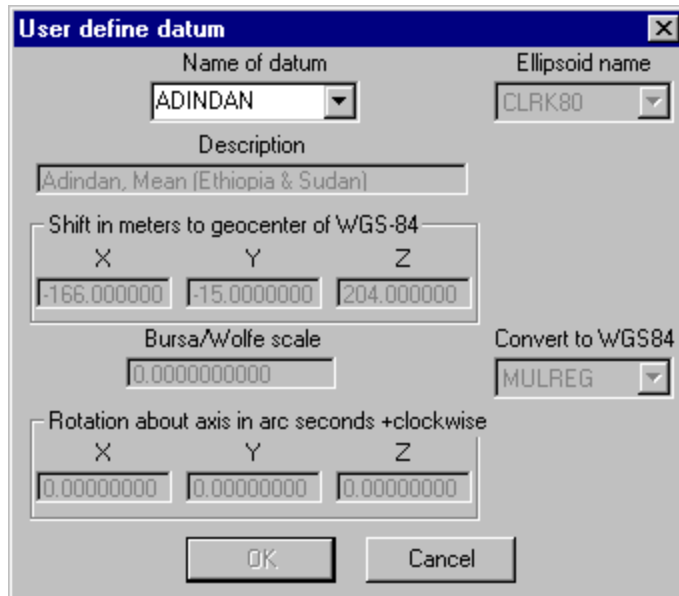
You can see a list of the available datums with their definitions or define your own datum using the User Defined Datum dialog box.

To define or view datums:

1. If you have not opened the *CoordCon* command bar:
 - Type **CoordCon** into the command bar.
 - OR
 - Click COGO, Utilities, and Coordinate Conversions.
 - The CoordCon command bar appears.
2. Click Coordinate Systems, and click Search. *The Search Coordinate Systems* dialog box appears.



3. Click Define Datum. The *User Defined Datum* dialog box appears.



4. To select an existing datum, click the Name of Datum box, and click the name of the datum.
5. To define a new datum:

- a. Type the name of the datum you want to define in the Name of Datum box (up to 22 alphanumeric characters).
- b. Type a description for the new datum into the Description box (up to 64 alphanumeric characters, including spaces).
- c. Click the Ellipsoid Name box, and click the name of the ellipsoid on which your datum is based.
- d. Click the Convert to WGS84 box, and click on the mathematical technique that should be used to convert geographic coordinates based on the datum being defined to WGS84 coordinates. You cannot add entries to this list, and you must select one of the entries in this list.

Molodensky	Molodensky Transformation, requires Delta X, Y, and Z
Bursa/Wolfe	a.k.a. Seven Parameter Transformation, requires Delta X, Y, & Z, and Rotation X, Y, & Z, and scale factor.
Multiple Regression	Defense Mapping Agency multiple regression formula. Requires .MRT files installed with Terramodel.
NAD27	Use the US National Geodetic Surveys NADCON program and data files. Includes Version 2 of Canadian National Transformation if data file is accessible.
HPGN	High Precision GPS Network, a.k.a. HARN (High Accuracy Reference Network); uses US National Geodetic Surveys NADCON program and data files.
WGS72	Uses analytical formula published by the Defense Mapping Agency, TR-8350.2-B.
NAD83	Equivalent to WGS84, no conversion necessary.
WGS84	Already referenced to WGS84, no conversion necessary.

- e. Type the parameters that apply to your datum into the appropriate boxes.
6. Click OK. The datum is ready for you to use when you define your coordinate systems.

Glossary of Geodetic Terms

This glossary describes terms used in this chapter that you might find unfamiliar or that are unique to **Terramodel**. The terms appear in alphabetical order.

Albers Equal-Area Conic Projection

A conic map projection that accurately preserves the areas of mapped objects. There is no distortion of shape or scale along two standard parallels, normally, or along just one. Parallels are unequally spaced arcs of concentric circles, more closely spaced at the north and south edges of the map. Meridians are equally spaced radii of the same circles, cutting parallels at right angles. This projection was developed by Albers in 1805.

American Polyconic Projection

This projection is so-named because the curvature of the circular arc for each parallel on the map is the same as it would be following the unrolling of a cone which had been wrapped around the globe tangent to the particular parallel of latitude, with the parallel traced on the cone. Thus, there are many (poly) cones involved. It is neither conformal or equal-area. Scale is true along each parallel and along the central meridian, but no parallel is "standard". It apparently was developed in 1820 by Ferdinand Hassler who was instrumental in establishing the U.S. Coast and Geodetic Survey.

Azimuthal Projection

A projection that accurately preserves the azimuth between certain objects. All projections introduce some distortion into the Cartesian results. To a certain degree, the mathematics chosen can eliminate certain types of distortion while introducing other types of distortion. An azimuthal projection preserves the azimuth between certain objects. Typically, such projections preserve only the azimuth of all points to a specific point, such as the origin of the coordinate system. In other words, on such a map, the direction from the origin to any other point on the map is shown accurately on the map.

Cartographer

A map maker or one who transfers and transforms positions of places on the surface of the earth to corresponding points on a flat sheet of paper, a map.

Central Meridian

The longitude of the horizontal center of a coordinate system. This longitude value is often the longitude origin of the coordinate system. In the case of the Transverse Mercator projection, the central meridian is the great circle at which the cylinder touches, or is tangent to, the earth.

Conformal Projection

A projection that can accurately preserve the shape of mapped entities with the drawback of scalar and area distortion. All projections introduce some distortion into the Cartesian results; to a certain degree, however, the mathematics chosen to perform the transformation can reduce certain types of distortion while introducing other types of distortion. A conformal projection preserves the proper angles between connected line segments and therefore tends to preserve the shape of geographic objects better than other types of projections.

Conic Projection

A projection based on the geometry of the cone. You can imagine a conic projection as similar to placing a party hat on the earth. The plain conic touches the earth at a specific parallel referred to as the Standard Parallel. Mathematically, the cone is often recessed into the earth to reduce the distance between the earth and the cone over the specific geographic area of interest. Reducing this distance reduces the distortion. Since the cone enters and then leaves the earth, two standard parallels result.

Coordinate System

An exact definition of a system of mathematics and geodetic constants that defines how a specific geographic location is converted to a set of two or three numbers, such as an x- and y-value (and possibly a z-value, also). In the cartographic context, most coordinate systems are Cartesian: the axes are orthogonal (perpendicular to each other) and the units are the same on all axes. The principal exception to this is the spherical coordinate system of latitudes and longitudes.

Cylindrical Projection

A projection based on the geometry of a cylinder. The spherical earth is mathematically projected onto a cylinder that surrounds the globe at a specific point. The traditional Mercator projection has the cylinder wrapped so that it is tangent to the earth at the equator. The Transverse Mercator projection places the cylinder tangent to the earth along the central meridian of the projection.

Datum

A datum is the combination of an ellipsoid, which specifies the size and shape of the earth, and a base point from which the latitude and longitude of all other points are referenced. Before satellites, lasers, and computer technology, establishing precise values for these points was impossible. More recently, however, many datums have been established and substantial amounts of data collected based on each. Data based on one datum will not necessarily overlay data based on another datum.

Eccentricity

Each ellipsoid that is used to represent the earth is flattened at the poles. Eccentricity is computed from the flattening factor via the formula $e^2 = 2f - f^2$.

Ellipsoid

The mathematical shape that best describes the shape of the earth. An ellipsoid is defined with two numbers: the major (equatorial) radius and the minor (polar) radius which is less than the equatorial radius. Therefore, a specific determination of the size of the earth is often referred to as an ellipsoid. For example, the phrase "Clarke ellipsoid of 1866" is frequently used to refer to the measurements of the size of the earth made by Clarke in 1866.

Equal Area Projection

A projection that accurately preserves the area of objects. All projections introduce some distortion into the Cartesian results; to a certain degree, however, the mathematics chosen can eliminate certain types of distortion while introducing other types of distortion. An equal area projection preserves the area of geodetic objects in that although the shape might be distorted, measuring the area of a geodetic object shows that accurate results have been produced. A coin placed on a map produced with an

equal area projection covers the same amount of real area no matter where it is placed on the map.

Equatorial Radius

The radius of the earth as measured at the equator.

False Easting

An element of a coordinate system definition. False easting is the value added to all x-coordinates to create a coordinate system with no negative x-values in the geographic region for which it is designed. Not all coordinate systems have a false easting.

False Northing

An element of a coordinate system definition. False northing is the value added to all y-coordinates to create a coordinate system with no negative y-values in the geographic region for which it is designed. Not all coordinate systems have a False northing.

Flattening Factor

This is the amount of flattening of the ellipsoid of the earth. It is computed as follows: $f = 1 - \text{Polar Radius} / \text{Equatorial Radius}$ and is approximately 1/300. For the GRS80 ellipsoid that NAD-83 uses, it is 1/298.257, for example.

Geodesy

The study of the size and shape of the earth and the mathematics involved in accurately modeling the earth. Geodesy also studies such phenomena as the variations in the shape of an imaginary ocean covering the entire earth. The variations in it would be caused by the non-uniform nature of the earth.

Geodetic

Of or relating to the study of the shape and size of the earth. A geodetic coordinate is the specification of a precise location on the surface of the earth.

Geodetic Reference System

A synonym for a datum. The combination of an ellipsoid, which specifies the size and shape of the earth, and a base point from which the latitude and longitude of all other points are referenced.

Geoid

The exact shape of sea level, as opposed to the ellipsoid model. The ellipsoid model is only an approximation required by the mathematical techniques of cartographers. The true shape of the earth (that is, the shape of the sea-level surface) varies from the ellipsoid model by as much as 80 meters above and 60 meters below the best fitting ellipsoid. This varying shape is caused by the non-homogeneous nature of the earth. When dealing with elevations, it is important to know if the elevation is referenced to the ellipsoid (a mathematical model of the size and shape of the earth) or to the geoid (the true representation of the size and shape of the earth).

Hotine Oblique Mercator Projection

This projection is used primarily in areas that do not have either a north-south or an east-west orientation. It is an oblique (slanted) cylindrical, conformal projection. Two of its meridians, 180 degrees apart are straight lines; the others are complex curves. Scale on the spherical form is true along the chosen central line or along two straight lines parallel to the central line. It was developed in 1946-47 by Martin Hotine.

International Survey Foot

Under this definition of the foot, its length is defined to be exactly 0.3048 meters. See the U.S. Survey foot for another definition of the length of a foot.

Lambert Conformal Conic Projection

This is a conic and conformal projection. The scale is true along two standard parallels, normally, or along just one. Meridians are equally spaced radii of the same circles, thereby cutting parallels at right angles. It is the most widely used projection for state plane coordinate systems and especially in areas that are predominantly oriented east-west. It was developed by Lambert in 1772, but was not widely used until it became the prototype for the first state plane coordinate system developed in North Carolina in 1933.

Latitude

The north/south component of the spherical coordinate system most widely used to record geodetic locations. Originally, when the earth was thought to be spherical, a degree of latitude represented one degree of arc on the surface of the earth, referenced to the center of the earth. Now that we know the earth to be ellipsoidal in shape, there are several types of latitude. The usual definition of latitude is the angle a line perpendicular to the surface of the ellipsoid makes with the plane of the equator. This is also referred to as the geographic latitude or geodetic latitude. Whenever the unqualified term latitude is used in **Terramodel**, you can safely assume that it refers to the geographic latitude. **Terramodel** requires that north latitudes be given in degrees, where positive numbers indicate north latitudes and negative numbers indicate south latitudes.

Longitude

The east/west component of the spherical coordinate system most widely used to record geodetic locations. Lines of longitude are great circles, all of which pass through the north and south pole and intersect the equator. That is, all lines of longitude proceed in a true north/south direction. The imaginary lines of longitude are assigned values which represent, in degrees of arc, the distance of the line from the prime meridian. The prime meridian is the line of longitude that passes through Greenwich, England.

Mercator Projection

This projection is cylindrical and conic with meridians being equally spaced straight lines. Parallels are unequally spaced straight lines, closest near the Equator and cutting meridians at right angles. Scale is true along the equator or along two parallels equidistant from the equator. It is used widely for navigation and was developed by Gerardus Mercator in 1569.

Meridian

In a cartographic/geodetic context, a meridian is a line of longitude. Meridian originally meant midday; a sailing vessel would determine its longitude by recording the time shown by the ship's chronometer when the sun reached its meridian (midday or noon). Because the chronometer always showed the time at Greenwich (the prime or zero meridian), the difference between the local meridian time (noon) and Greenwich time was the vessel's longitude.

NADCON Software

A computer program developed by National Geodetic Survey to convert latitudes and longitudes from NAD-27 to NAD-83. NAD is an abbreviation for North American Datum (see below).

North American Datum of 1927 (NAD-27)

A datum based on the Clarke ellipsoid of 1866, with a base station at Meades Ranch in Kansas. NAD-27 used the latitude and longitude for Meades Ranch and the Clarke 1866 earth-based datum values to determine the latitude and longitude of over 50,000 surveying monuments throughout the contiguous United States and Alaska. The monuments, often simply brass spikes driven into the ground, served as starting points for more local surveying and mapping efforts. Over the years, stations were added to the network and there are now over 267,000 stations of varying quality. Use of this datum is gradually being replaced by the North American Datum of 1983.

North American Datum of 1983 (NAD-83)

An earth-centered datum based on the Geodetic Reference System of 1980. The size and shape of the earth was determined through measurements made by satellites and other sophisticated electronic equipment; the measurements accurately represent the earth to within two meters. The datum is called earth-centered (or geocentric) because its reference point is the center of the earth, as opposed to a point on the earth's surface. In developing NAD-83, the National Geodetic Service used data from NAD-27. As a result, the latitude and longitude assigned to all 267,000 NAD-27 monuments has changed by as much as 350 feet horizontally.

Origin Latitude

The latitude of the geodetic location at which the projection formulae produce zero values for the resulting Cartesian coordinates, before the false easting and false northing are applied. In certain cases, the origin latitude is implied by other elements of the coordinate system definition. The origin latitude of the Hotine Oblique Mercator, for example, is a point where the central great circle intersects the equator and it is calculated from the other elements of the coordinate system definition.

Origin Longitude

The longitude of the geodetic location at which the projection formulae produce zero values for the resulting Cartesian coordinates, before the false easting and false northing are applied. In many coordinate system definitions, the origin is implied by other elements of the definition. For example, the origin longitude of all coordinate systems based on the Transverse Mercator projection is the central meridian, simply because it is the nature of the projection. Coordinate systems based on any conic projection, however, require that you specify both an origin longitude and an origin latitude.

Parallel

In a cartographic/geodetic context, a parallel is a specific line of latitude. Lines of latitude are called parallels because they are parallel to all other lines of latitude.

Projection

A mathematical development of one surface onto another. In the cartographic context, one surface is the earth and the other is a flat surface such as a map. The mathematics usually convert from the spherical coordinates of latitude and longitude to a Cartesian coordinate system. The projection is one element of a coordinate system definition.

Scale of the Central Meridian

See Scale Reduction Factor.

Scale Reduction Factor

The degree of distortion produced by a cylindrical projection can be reduced by shrinking the cylinder into the earth slightly. This reduces the greatest distance from the surface of the earth to the cylinder over the geographic region of interest, thus reducing the maximum degree of distortion. This also distributes the distortion more evenly across the map. Mathematically, this shrinkage is achieved by applying a scale factor to the cylinder. This scale factor is often called the scale reduction factor or the scale of the central meridian. This value is always close to but less than one. In fact, it is usually set to the cosine of one quarter of the width of the geographic region being mapped. For example, a UTM zone is six degrees wide, therefore the scale reduction factor is set at the cosine of 1.5 degrees, or 0.9996.

Standard Parallel

An element of conic projections indicating the latitudes where the projection cone enters and leaves the earth. For a coordinate system in the northern hemisphere, for example, the northern standard parallel indicates where the cone enters the earth, and the southern standard parallel indicates where the cone leaves the earth. It might help to imagine the conic projection as a conic party hat placed on top of the earth. Shrinking the cone into the earth reduces the maximum distortion introduced by the projection and the projection distortion is more evenly distributed over the region being mapped. The standard parallels, therefore, indicate the region being mapped, and also, because of the distance between the two, the degree to which the cone has been mathematically recessed into the earth. Some coordinate systems based on conic projections have a single standard parallel, indicating that the cone has not been recessed into the earth.

State Plane Coordinate Systems

A complete system of coordinate systems by which Cartesian coordinates are assigned to all 50 states. The system consists of as many zones as are necessary to define coordinate systems where the distortion introduced by the projection is less than one part in 10,000. To achieve this objective, different states use different projections, and most states require more than one zone. For example, the state of Idaho, whose largest extent is north/south, uses the Transverse Mercator projection, and because of its size, requires three separate zones to achieve distortion of less than one part in 10,000 within any zone. (Alaska requires 10 zones to maintain this accuracy.)

Universal Transverse Mercator (UTM)

A series of 120 coordinate systems based on the Transverse Mercator projection originally developed by the U. S. Army for a world-wide mapping project. Sixty zones are used to map the northern hemisphere and the remaining 60 apply to the southern hemisphere. Each zone is six degrees wide and is numbered. Zone one covers longitudes of 180W through 174W. The remaining zones are numbered sequentially as they move east. All zones have their origin at the equator, use the meter as the system unit, and have a false easting of 500,000 meters and a false northing of zero. A scale reduction factor of 0.9996 is used on all zones. Zones for the Southern Hemisphere are identical to their northern counterpart except that the false northing is set to 10,000,000 to eliminate negative y-coordinates. The military also uses a system of lettering to identify each 100,000 meter square in each zone, with the first letter of a pair identifying a column and the second letter identifying the row.

U.S. Survey Foot

In 1959, the Foot System was redefined by agreement among the nations that use the system. This reduced the length of the existing United States Foot System by approximately 2 parts in 1,000,000. Under this agreement, the foot was redefined by the value of 39.37 inches = 1 International Meter. Using this equation, the U.S. (or American) Survey Foot is equal to 0.3048006096012.. meters.

World Geodetic Reference System of 1984 (WGS-84)

The United States Defense Mapping Agency's Datum. This datum is a global datum based on electronic technology which is still to some degree classified. Data on the relationship of as many as 65 different datums to WGS-84 is available to the public. As a result, WGS-84 is becoming the base datum for the processing and conversion of data from one datum to any other datum. It should be noted that the difference between WGS-84 and NAD83 is very minor. For example, a distance using WGS84 datum coordinates was computed from a point in Mount Wilson, California to the Empire State Building in New York and when compared to the distance when using NAD83 datum coordinates for the same points, the difference was less than 1 mm.

Chapter Fourteen

Creating Reports

This chapter explains how to create reports with the P3Pad Report Editor.

In Terramodel, you can quickly and easily generate reports for your project, automatically formatted appropriately for the type of report you select. Edit your reports with the P3Pad Report Editor, adding notes and explanations where you want. Then, add graphics or change fonts and font styles to give your printed reports impact and a professional appearance.

This chapter covers the editing and formatting features of the P3Pad Report Editor. Because Terramodel can generate a wide variety of reports, for information about a specific report type refer to the online help for the associated command.

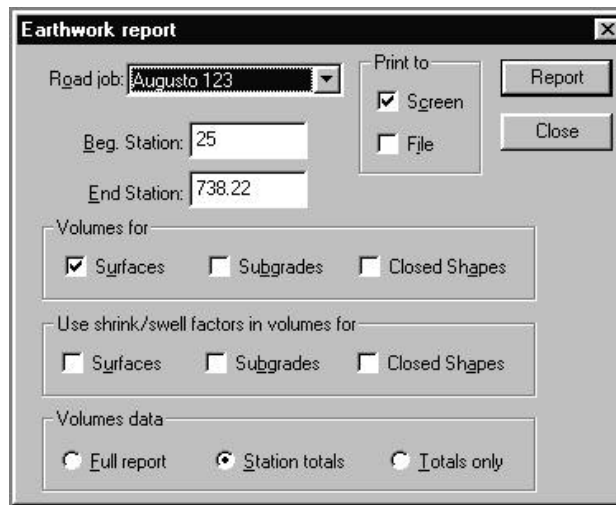
The P3Pad Report Editor

The P3Pad Report Editor is a text processor similar to the Microsoft® WordPad. In addition to basic text entry and editing commands it includes features for inserting graphics, and for formatting text:

- Insert formatted date and time stamps
- Create and insert new graphics and other objects
- Insert existing files
- Set the font, size, style, and color for selected text
- Set indents for selected paragraphs
- Set or clear Tab stops
- Preview your report before printing

Creating a Report

The Print to Screen option on a report dialog box such as the Earthwork Report dialog box, shown in the following graphic, creates and opens the report in P3Pad for editing, formatting, and printing.



To create a report in P3Pad

- On the dialog box for the specified report, click the Print to Screen check box, then click Report.
P3Pad starts and displays the report.

Saving a Report

You have two methods for saving a report file. First, on report dialog boxes you can select to print the report to a file as well as to the screen. Second, within P3Pad, use the Save command to save the formatted report to disk, or Save As to save the report with a different file name.

Print to File

If you select the Print to File check box on the report dialog box, the report is saved as an ASCII text file with the name and location you specify. You can open the report file in other applications.

To print the report as a file

1. On the report dialog box, click the Print to File check box.
2. Click Report.
3. On the Report File dialog box, type the name you want for the report file and select the location folder.
4. Click Save.

Saving a Formatted Report

The Print to File option does not save any formatting included with the report. To save formatting, use the **Save** command from within P3Pad. The first time you save the report, type a file name on the *Save As* dialog box. When you save the file after that, P3Pad saves the file to disk using that file name.

To save a report with formatting

1. On the report dialog box, select the Print to Screen check box.
1. Click Report.
2. Edit and format the report document, using the procedures described in the following sections.
3. From the P3Pad FILE menu, select Save. The *Save As* dialog box appears.
4. Type the file name for the report and select the location folder.
The default file type is a WordPad document with the file extension .DOC. You can also save the file in a Rich Text Format (.RTF) if you prefer.
2. On the dialog box, click Save.

To save a report with a new name

1. From the P3Pad FILE menu, click Save As. The *Save As* dialog box appears.
3. Type the new file name for the report and select the location folder.
The default file type is Rich Text Format with the file extension .RTF.
4. Click Save.

Editing Report Text

Adding Text

After creating the report, use the keyboard to type notes or explanations wherever you need them. Use standard word processing procedures to position the pointer in the report where you want to enter text.

Selecting Text

To delete, move, copy, or format text you must first select the text you want to change. P3Pad provides several ways to select text. Selected text is highlighted.

To select a block of text

- ▶ Click and drag the pointer across the text.

To select a word

- ▶ Double-click the word.

To select an entire line

- ▶ Move the pointer to the left margin, then click.
The entire line is selected.

Deleting Text

To remove selected text select the Cut command, the Back Space key or the Delete key. When you use Cut, the deleted text can be pasted elsewhere in the report, in another document, or in the original place to undo the cut. If you use the Back Space or Delete keys, the text is permanently deleted and cannot be pasted in the report.

To delete text from the report

1. Select the text you want to remove.
2. From the EDIT menu, click Cut.
OR
Press Ctrl+X.

Copying Text

Use the Copy command to paste the selected text elsewhere in the report without deleting it. You can paste the same text in multiple places in the document.

To copy text within the report

1. Select the text you want to copy.
2. From the EDIT menu, click Copy.
OR
Press Ctrl+C.
3. Click the pointer at the place in the report where you want a copy of the text to appear.
4. From the EDIT menu, click Paste.
OR
Press Ctrl+V.

Moving Text

To move text within the report

1. Select the text you want to move.
2. From the EDIT menu, click Cut.
OR
Press Ctrl+X.
3. Click the pointer at the new location in the report.
4. From the EDIT menu, click Paste.
OR
Press Ctrl+V.

Undo

Reverse the action of the previous editing or formatting command with Undo on the Edit menu. **Undo** is not available for all commands, and reverses only the most recent command.

To reverse the previous command

- From the EDIT menu, click Undo.
OR
Press Ctrl+Z.

Finding and Replacing Text

P3Pad includes Find and Replace functions you can use to find specified words or phrases, or to replace specified words throughout the report.

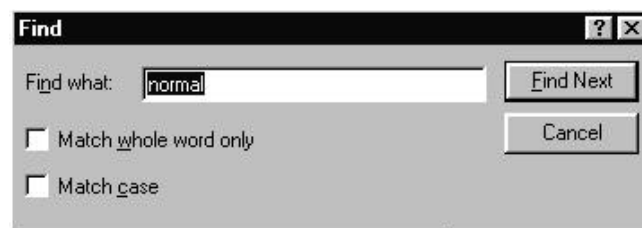
Find and Find Next

The Find command locates and selects the word or words you enter on the Find dialog box. Find Next searches for the next instance of the specified text.

By entering identifying information you can use Find as a “Go To” function to navigate long reports. For example, on a Roadway report, enter the numbers for a station to display the appropriate section.

To Find text in the report

1. From the EDIT menu, click Find.
OR
Press Ctrl+F.
The *Find* dialog box appears.



2. In the Find What text box, type in the word or phrase you want to find. Type over any previously entered text if it is not the text you want to find.

If you select text in the report before using the Find command, the selected text appears in the text box.

3. Click the Match Whole Word Only check box to prevent selection of text included in longer words.
For example, if you type “form” and select Match Whole Word Only, the search finds “form” and not “information.”
4. Select the Match Case check box to locate words that exactly match the capitalization you use when typing the word.
For example, if you type “Form” and select Match Case, the search finds “Form” but not “form,” “FORM,” or “information.”

- Click Find Next to locate the next occurrence of the word in the Find What text box.

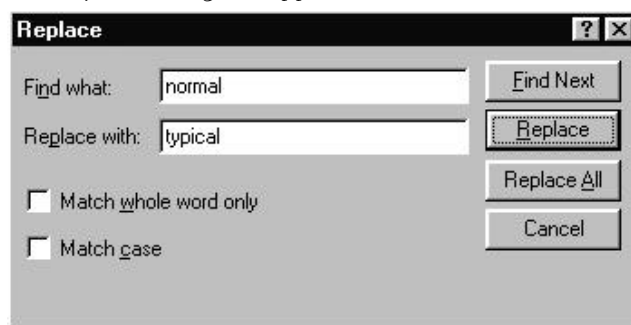
If P3Pad finds text matching your entry, that part of the report displays with the matching text selected. A message appears if the report editor cannot find a match.

Replacing Text

With the Replace command, you can locate specified text and replace it with other text, either one occurrence at a time or all occurrences at once.

To find and replace text

- From the EDIT menu, click Replace.
OR
Press Ctrl+H.
The *Replace* dialog box appears.



- In the Find What text box, type in the word or phrase you want to find. Type over any previously entered text if it is not the text you want to find.

If you select text in the report before using the Find command, the selected text appears in the text box.
- In the Replace With text box, type the replacement word or phrase.
- Select the Match Whole Word Only check box to prevent selection of text that might be part of longer words.

For example, if you type “form” and click Match Whole Word Only, the search finds “form” and not “information.”

- Select the Match Case check box to locate words that exactly match the capitalization you use when typing the word.

For example, if you type “Form” and click Match Case, the search finds “Form” but not “form,” “FORM,” or “information.”

6. Click Find Next to locate the next occurrence of the word in the Find What text box.

If P3Pad finds text matching your entry, that part of the report displays with the matching text selected. A message appears if the report editor cannot find a match.

7. To replace the selected occurrence of the text, click Replace.
OR
To skip this occurrence and not replace the text, click Find Next.
OR
Click Replace All to replace the text in all occurrences in the report.
8. When you complete replacing text, click the Close button on the dialog box.

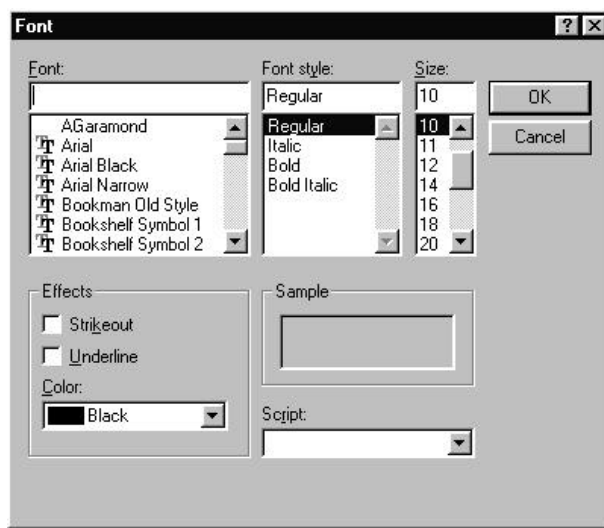
Formatting the Report

P3Pad contains basic formatting controls to sharpen the appearance of your document, add emphasis by changing text styles or color, or fit the report on a page by changing text size. Make format selections from dialog boxes you activate with menu commands or select controls directly by clicking buttons on the Format bar.

If the data columns do not line up properly after you generate a report, check the font used by the template. To properly align columns, try selecting a mono-spaced font such as Courier New for the report text.

Text Formatting with the Font Dialog Box

The *Font* dialog box displays all the settings options available for formatting selected text. From the dialog box you can select the font and size, apply styles, and set text color all at once. You can apply *Font* dialog box formatting to selected letters or words, or apply format settings to the entire report by using the Select All command on the Edit menu.



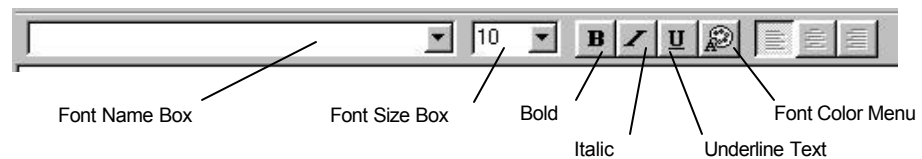
To format text from the Font dialog box

1. From the FORMAT menu, click Font.
The *Font* dialog box appears.
2. Type the font name in the Font box.
OR
Select a font name from the list.
3. Type the style in the Font Style box.
OR
Select a style from the list.
4. Type a font size in the Font Size box.
OR
Select a size from the list.
Fonts must be at least 2 points high, and can be fractional, such as 10.5.
5. Click the Strikeout check box to add a hyphen across each letter in the selected text.
6. Click the Underline check box to add underlining to the selected text.
7. From the Color list, click the color you want to apply to the selected text.
8. Click **OK** to close the dialog box.

Text Formatting from the Format Bar

The Format bar displays buttons for most of the commands you need to format report text. You can set the font, font size, font style and color for selected text. Left-align, right-align, and center text controls are not available in P3Pad.

You can apply Format bar formatting to selected letters or words, or apply format settings to the entire report by using the Select All command on the EDIT menu.



To select a font

- From the Font list, click the font name you want to use.

To set a font size

- From the Size list, click the font point size.
OR
Type the point size you want in the Size text box, then press Enter.

To apply a bold, italic, or underlined style to text

- Click the buttons for the styles you want to apply.


You can apply more than one style to the selected text.

To remove bold, italic, or underlined style from text

1. Select the text formatted with the style you want to remove.
The button for the applied style appears selected.
2. Click the style button to cancel the selection.
The style is removed from the text.

You can remove one style at a time from selected text when more than one style has been applied.

To apply a color to selected text

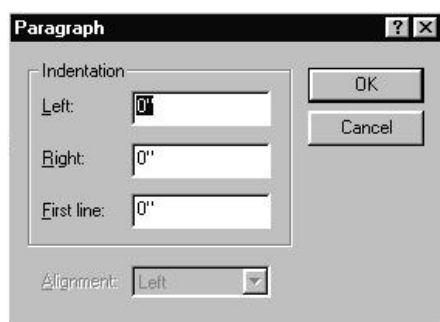
1. Click .
2. From the menu, click the color you want to apply.

To control the Format bar display

- From the VIEW menu, click Format Bar to switch between removing or displaying the Format bar.

Setting Paragraph Indents

Set left, right, and first line indents for selected paragraphs on the *Paragraph* dialog box. To set a single paragraph, click the insertion point anywhere within the paragraph. You do not need to select all the paragraph's text. To apply the same indentation settings to the entire report, use the Select All command on the EDIT menu.



To set paragraph indents

1. Select the paragraphs you want to format.
2. From the FORMAT menu, click Paragraphs.
3. Enter an indent value, in inches, in the Left, Right, and First Line boxes.
4. Click OK to apply the indents.

Setting and Clearing Tabs

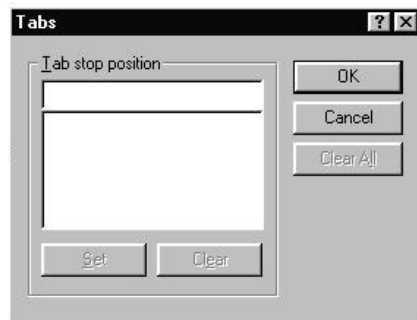
When you create a report, P3Pad uses a template appropriate for the report type to format the project data. Because data columns are already set in the template, it is unlikely that you will need to set tab stops for the generated data.

If the data columns do not line up properly after you generate a report, check the font used by the template. To properly align columns, try selecting a mono-spaced font such as Courier New for the report text.

P3Pad includes default tab stops at half inch intervals from the left margin. To set tab stops in other positions to align data and notes you add to the report, you can set tab stops with the *Tabs* dialog box.

When you set a tab stop, you automatically clear all default tab stops to the left of your entry, although they are still active to the right of your setting. For example, if you type 2.25 on the *Tabs* dialog box, the first tab stop for the selected paragraph is

2.25 inches from the left margin. The next tab stop is at 2.5 inches and every following half inch.



To set a tab stop

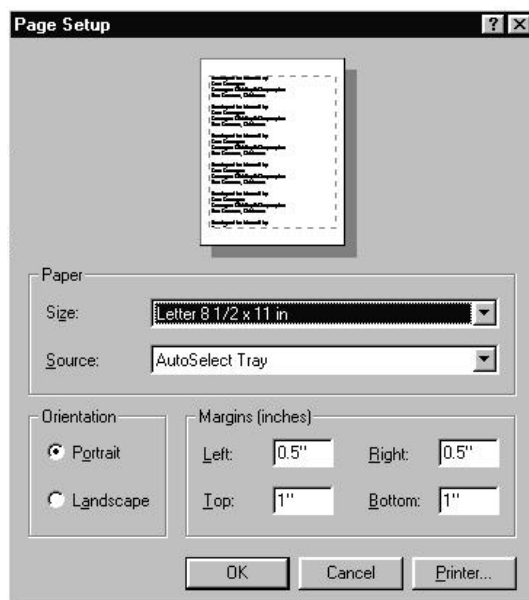
1. Select one or more paragraphs in the report you want to set with tab stops.
2. From the FORMAT menu, click Tabs.
3. In the Tab Stop Position box, type the position of the tab stop in inches from the left margin.
4. Click Set. The position is added to the Tab Stop Position list.
5. Click OK to close the dialog box.

To clear a tab stop

1. Select the paragraphs you want to clear of the tab stop.
2. From the FORMAT menu, click Tabs.
3. In the Tab Stop Position list, click the tab stop you want to clear.
4. Click Clear. The selected tab stop is removed from the list.
5. Click OK to close the dialog box.

Setting Page Layout

The report template includes default settings for margins, paper size, and paper orientation. You can change these settings on the *Page Setup* dialog box.



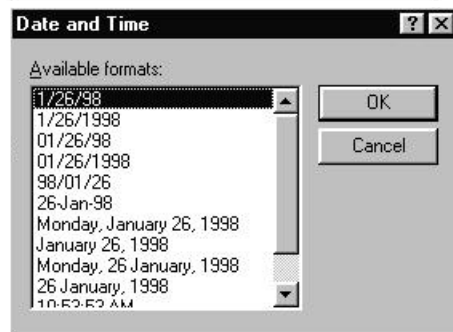
To enter page layout settings

1. From the FILE menu, click Page Setup.
2. From the Size list, click the paper size you want to use.
3. From the Source list, click the source tray or manual feed for your printer.
4. In the Margins text boxes, type values in inches for the left, right, top, and bottom margins.
5. Click Landscape to print the report along the length of the paper.
6. To enter printer settings, click Printer.
The *Page Setup* printer dialog box appears with printer setup information.
Repeat steps 2-6 to Select another printer.
Click Properties to enter other printer settings.
Click OK to close the *Page Setup* printer dialog box.
7. Click OK to close the *Page Setup* dialog box.

Inserting Objects into the Report

Inserting the Date and Time

The Date and Time dialog box provide you with several predefined formats for entering the current date and time setting on the computer's system clock. You might use the Date and Time dialog box to time stamp a note you add to a report file you previously saved.



To add the date and/or time to a report

1. Click the pointer on the report where you want to place the date or time.
2. From the INSERT menu, click Date and Time.
The *Date and Time* dialog box appears.
3. From the Available Formats list, click the format you want to use.
4. Click OK.
The current date and/or time is added to the report.

Inserting Objects

P3Pad supports Object Linking and Embedding (OLE) technology, a way to share data between different applications. That means that from the Insert Objects dialog box you can select from a list of object types, including graphics files, word processor documents, spreadsheets, tables, presentations—even sound and video files, to enhance your report.

You have three methods of sharing data between P3Pad and other applications:

- Create a new object from the *Insert Objects* dialog box.

- Select an existing file to insert from the *Insert Objects* dialog box.
- With Paste Special, copy linked data from an open application.

Inserting New Objects

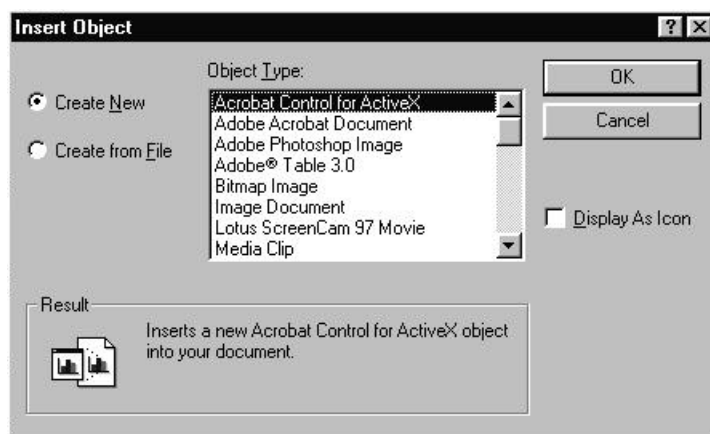
When you select Create New objects, a window to the selected application, if it is installed on your computer, is embedded in the report. Create the object in this window using the menus and commands available in the application.

For example, if you insert a new Microsoft Excel spreadsheet object, you can enter formulas and data, format the spreadsheet, and access commands exactly as in Microsoft Excel.

If you view or distribute the report in electronic format, you can insert an icon linked to document file instead of the actual document.

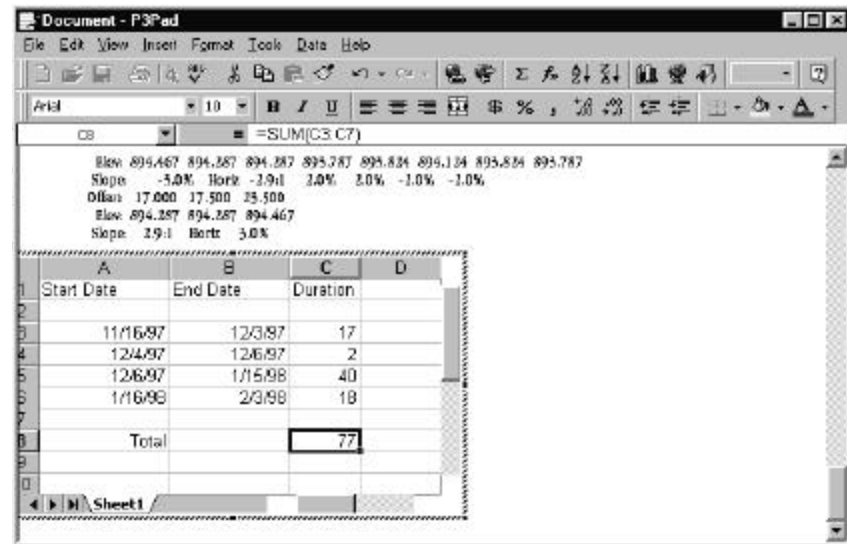
To insert a new object

1. Click the report where you want the new object to appear.
2. On the INSERT menu, click Insert Objects.
The *Insert Objects* dialog box appears. The default selection is Create New.



3. From the Object Type list, click the object you want to insert.
4. Click OK.
A new document window for the object application is inserted into the report. Menus and toolbars for the object application appear in the report editor while the window is active.

If the selected application is not installed on your computer, a warning message appears.



5. Complete the object document in the window.
6. Click the report anywhere outside the object window.
The report document is now active, P3Pad menus and commands are available, and the object document displays and prints as a seamless part of the report document.

To edit an object

1. Click the object you inserted within the report.
The object displays in the object application window and the object menus and commands are available.
2. Make the changes, additions, or deletions to the object.
3. Click the report anywhere outside the object window.

To insert a new object as an icon

1. On the *Insert Objects* dialog box, make selections to create a new object, using the steps listed in the preceding section.
2. Click the Display as Icon check box.
An image of the icon appears on the dialog box below the check box.
3. Click OK.
The object application starts and opens a new document window.

If the selected application is not installed on your computer, a warning message appears.

4. Create the document in the object application, save the document, and close the object application.
An icon for the object application appears on the report at the point you selected to insert the object.
5. Double-click the icon to start the object application and display the document.

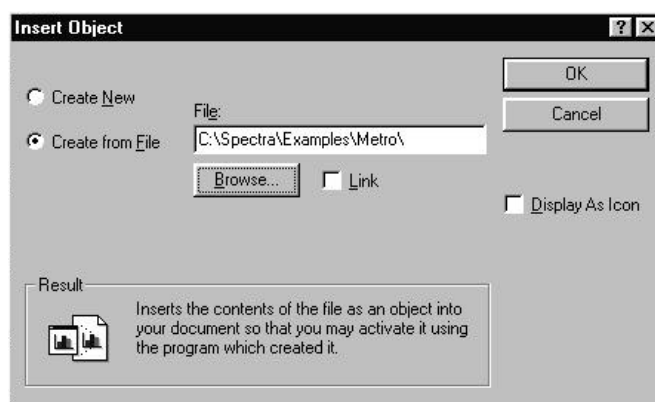
Inserting Existing Files

From the Insert Object dialog box you can select existing document files to insert into the report. When you insert an existing file, you have three options for displaying the contents.

- Embed the file as an object you can view and edit within P3Pad.
- Display an icon of the original file. Selecting the icon starts the object application and opens the document.
- Embed a linked image of the file contents. This image reflects any changes you make to the original file, however you cannot edit the object from within P3Pad.

To embed a file as an object.

1. Click the report where you want the new object to appear.
2. On the INSERT menu, click Insert Objects.
The *Insert Objects* dialog box appears.



3. Click Create from File.
4. In the File box, type the path and file name for the file.
OR
Click Browse to select the file from the *Browse* dialog box.

5. Click OK. The document appears in the report, displayed within a bounding outline.
6. Click the report anywhere outside the object window.
The report document is now active, P3Pad menus and commands are available, and the object document displays and prints as a seamless part of the report document.

To edit an object created from a file

1. Double-click the object you inserted within the report.
The object displays in the object application window and the object menus and commands are available.
2. Make the changes, additions, or deletions to the object.
The embedded object is a copy of the file. Any changes you make to it within P3Pad are not reflected in the original.
3. Click the report anywhere outside the object window to close it and make the report active again.

To insert an existing file as an icon

1. On the *Insert Objects* dialog box, click Create from File. Select the file.
2. Click the Display as Icon check box.
An image of the icon for the file type appears on the dialog box below the check box.
3. Click OK.
An icon for the object file type appears on the report at the point you selected to insert the object.
4. Double-click the icon to start the object application and display the document.

To insert an object as a linked file

1. Click the report where you want the new object to appear.
2. On the Insert menu, click Insert Objects.
The *Insert Objects* dialog box appears.
3. Click Create from File.
4. In the File text box, type the path and file name for the file.
OR
Click Browse to select the file from the *Browse* dialog box.
5. Select the Link check box.
6. Click OK.
A copy of the linked file appears within a bounding outline on the report at the point you selected to insert the object.

- To make changes on a linked file, open the file in the application in which it was created. Make the changes, and then save the file.

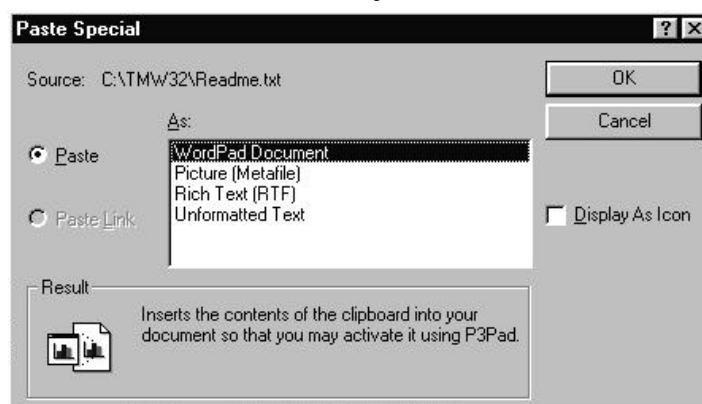
Changes to the original file appear in the copy embedded in the report.

Inserting with Paste Special

Use Paste Special to share data between a source document open in an application and a report in P3Pad. You can use Paste Special to copy the source material multiple times in the report.

To use Paste Special

- In Terramodel, create the report.
- Start the application for the source document and either open an existing source document or create a new source document.
- Copy the text, data, or graphic in the source application you want to include in the P3Pad report. See the documentation for the source application if you aren't sure how to use the application.
- Minimize the source application and make P3Pad the active application.
- In P3Pad, click the report where you want to insert the material from the source document.
- On the EDIT menu, click Paste Special.



- Click Paste to insert a copy of the source material.
OR
Click Paste Link to insert a copy of the source material that is linked to the original file.
- Click the file type from the list.

Printing the Report

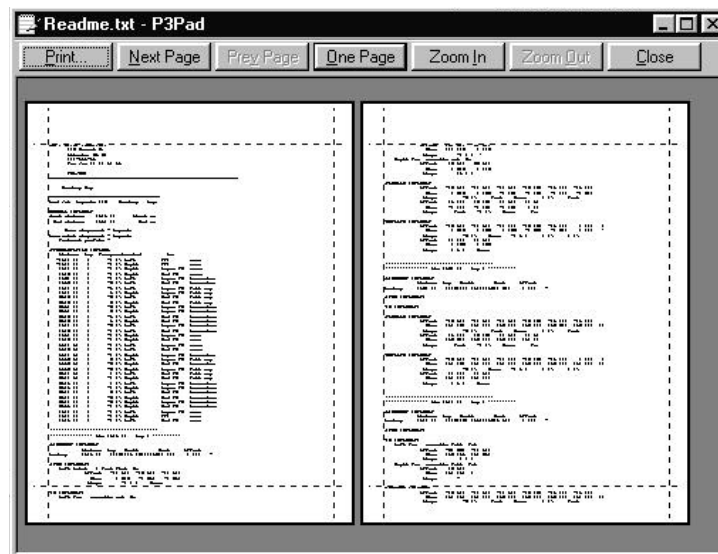
To complete your report after you finish editing and formatting it, preview how it will look, and then print a copy of the document.

Print Preview

Select Print Preview to check your report formatting before you send it to the printer.

To display the Print Preview screen

- From the FILE menu, click Print Preview.
The document appears in the Print Preview window with dotted lines marking the margins.



Print Preview Controls

- Click Next Page to view the next page.
- Click Prev Page to return to the previously displayed page.
- Click Two Page to display two pages side-by-side.
- Click One Page to display the report one page at a time.

- Click Zoom In to magnify the image size.
Print Preview has three image magnification sizes.
Selecting Zoom In automatically sets to single page display.
- Click Zoom Out to reduce the image size.
Using Zoom Out to return to the smallest image size automatically restores the view to two-page if you previously selected that mode.
- Click Close to close the Print Preview window.
- Click Print to begin printing the document.

Printing the Report

Selecting Print on the File menu or on the Print Preview window displays the *Print* dialog box. From the dialog box you can select another printer, set the number of copies, and specify the pages you want to print. Properties gives you access to Windows printer settings. Click OK to begin printing.

Chapter Fifteen

Terramodel Visualizer

Introduction

Terramodel Visualizer is an interactive, project-visualization tool that displays your Terramodel project as a three-dimensional model. You can interactively move to any vantage point in your project and view it under any lighting conditions you specify.

The basic Terramodel Visualizer module includes the tools to:

- Position and fly through a project
- Illuminate the project
- Control shaded surface display modes
- Add three-dimensional representations of landscaping features (e.g. trees)
- Record flight paths through the project
- Capture and print images
- Make AVI movies

Optional modules allow you to:

- Edit Terramodel project points without leaving Terramodel Visualizer

- Drape orthographic images over the three-dimensional surfaces
- View your project in stereo with specialized glasses, graphics card and emitter, using a single monitor

Terramodel Visualizer Documentation

This chapter is a brief guide to the installation of Terramodel Visualizer, and overview of Terramodel Visualizer's capabilities. Included as an appendix to the User Guide, is a section that describes the tree, shrub and geotypical texture images available as well as examples of Terramodel Visualizer's capabilities.

Online Help

The primary source of documentation for Terramodel Visualizer is the online help system. When you are using Terramodel Visualizer, press F1 or click HELP and Contents to access the help system.

Installing Terramodel Visualizer

Minimum Requirements

Terramodel Visualizer can run on any machine that can run Terramodel, however, visualization is extremely demanding on the computer processor, memory, and your graphics adapter. For huge data sets, the fastest processor and graphics adapter available combined with the maximum amount of memory is recommended.

Installing Terramodel Visualizer

To install Terramodel Visualizer, put your autorun Terramodel CD in the CD drive. Then follow the setup instructions that appear on your screen. When requested, select Terramodel Visualizer as the program that you want to install.

Do not install Terramodel Visualizer in the same directory where you installed Terramodel. Note that Terramodel Visualizer 1.0 cannot read Terramodel Visualizer 2.0 files, however Terramodel Visualizer 2.0 can read and update Terramodel Visualizer 1.0 files.

Uninstalling Terramodel Visualizer

To uninstall Terramodel Visualizer, click the Uninstall icon in the Terramodel Visualizer program group.

Maximum Number of Objects (p3server.ini) Setup

You may need to set the maximum number of objects in a project file for Terramodel Visualizer. Terramodel Visualizer does not use settings you entered in the p3server.ini file for Terramodel. To change the Terramodel Visualizer p3server.ini file:

1. Start NotePad, WordPad, or a text processor.
2. Click FILE, and click Open.
3. Locate the Terramodel Visualizer folder.
4. From the Terramodel Visualizer folder, select p3server.ini.
5. Under [DBTYPE], enter values for:
maxRec= # records - maximum number of objects for the current file
maxPt= point number - highest point number for the current file
Both values **must** be set to the same number and may be set the same as in Terramodel.
6. Save the file.

Starting and Using Terramodel Visualizer

You can start Terramodel Visualizer in a variety of ways:

- Double-click the Terramodel Visualizer icon in the program group created when you installed the software.
- Double-click the Terramodel Visualizer.exe file in Explorer.

Terminology

Environment - An **environment** is the Terramodel Visualizer work area displaying projects three-dimensionally. In addition to the project display, an environment contains information such as position and intensity of lights, position and orientation of cameras, and recorded paths.

Camera - A **camera** is the viewpoint you maneuver while in Flying Mode. An environment may contain an unlimited number of independently positioned and oriented cameras.

Rotating Mode - In **Rotating Mode**, you move the project around its center on any axis using either mouse or keyboard control.

Flying Mode - In **Flying Mode**, you move a camera over and around the project simulating forward flight using either mouse or keyboard control.

Path - A path is a recording of camera flying movements you can follow later and use in the generation of AVI movies.

The Terramodel Visualizer help system contains an extensive glossary of related terms.

Application Ideas for Terramodel Visualizer

Terramodel Visualizer is a versatile tool used to display photorealistic images from data points. While we cannot describe all the possible uses of this program, this guide lists some common applications, explains basic operations, and tells you how to get more help. Use the basic information we have provided here to invent your own unique applications.

Use Terramodel Visualizer to better understand and interact with 3- dimensional data. This data can be either existing or proposed conditions, or a combination of the two. 3-D visualization can help you understand the data better and find problem areas. Terramodel Visualizer also lets you interact with the data so that it can be viewed from any angle and any distance. This enhanced ability to better understand the data helps to eliminate errors, improve the communication of abstract data, and speeds the design process.

Use Terramodel Visualizer to produce prints, plots, and image files that are used to better communicate the shape of the data. Clients that collect, process, and design using topographic data can now produce another saleable product – a 3-D view. You can

boost your profits by producing photorealistic images that can easily be produced from original data.

Use Terramodel Visualizer to create sophisticated fly- or drive-through movies. Imagine how impressed your clients (or potential clients) will be when they see a photorealistic fly-through of their site.

There are many practical every day uses for Terramodel Visualizer, here are a few of the common applications:

- Bathymetry
- Scientific visualization –
- Environmental analysis
- Landfill design
- Site design
- Surveying
- Photogrammetry
- Landscape architecture
- Law enforcement
- Construction
- Military mission planning
- GIS
- News and weather animation
- Entertainment
- Forestry
- Fish and wildlife management
- Trip planning
- Real-estate sales and marketing

Using Terramodel Project Files in Terramodel Visualizer

Terramodel Visualizer is designed to work with Terramodel version 9.30 and later project files. To view older project files, open them in the current version of Terramodel and save the file.

Because of the computational requirements of visualization, Terramodel Visualizer requires more memory and computational resources than Terramodel with the same project file.

To maximize the speed of Terramodel Visualizer:

1. Use Terramodel to hide layers containing data that does not need to be visualized.
2. If desired, save this file to a new name, so your original settings in Terramodel are not affected.
3. Open Terramodel Visualizer 2.0. Click FILE and Preferences. The Preferences dialog box appears. Click the *Misc.* tab, and click the *Only Form Geometry from Visible Layers* check box to activate it.

Terramodel dramatically enhances the functionality of Terramodel Visualizer. Use Terramodel to import data from many sources, its ability to translate points between multiple coordinate systems and its DTM functionality, the uses of Terramodel Visualizer are greatly increased.

Learning How to Use Terramodel Visualizer

Once you have prepared your project in Terramodel, you are ready to view it in Terramodel Visualizer.

To learn how to view the project in Terramodel Visualizer:

1. Start Terramodel Visualizer, and press F1. The help system opens, displaying the “Getting Started with Terramodel Visualizer” topic. This topic gives you an overview of using Terramodel Visualizer.
2. Follow the instructions in the “Getting Started with Terramodel Visualizer” topic. When you need more detailed instructions, click one of the jumps within the topic. Click Back to return to the “Getting Started with Terramodel Visualizer” topic.

3. Click FILE and Exit to close the Help system.

Now that you have reviewed the basic steps for using Terramodel Visualizer, we recommend that you run Terramodel Visualizer and look at several of the example files. Some of the example files include features that are available only in optional modules.

Example of Following a Recorded Path

Open the mountain road project and follow the recorded path to see how Terramodel Visualizer renders large road projects.

1. Click the Terramodel Visualizer shortcut on the Start menu. Terramodel Visualizer starts with an empty environment.
2. Click FILE, and click Open. The Open dialog box appears.
3. In the dialog box, open the EXAMPLES folder within the Terramodel Visualizer folder. EXAMPLES contains four folders: GOLF, LANDFILL, METRO, and MOUNTAIN.
4. Click MOUNTAIN. The MOUNTAIN folder contains the project and environment files.
5. Click the environment file, MOUNTAIN.EN. Terramodel Visualizer opens the environment containing the mountain road project.
6. Move the camera into position for the fly-through by clicking Follow Path from the Navigation menu.
7. Select a flying speed by pressing a number key from 1 (slowest) to 9 (fastest). The camera moves forward along the path, flying above the roadway. Press different number keys to vary the speed, or press 0 (zero) to stop. When the camera reaches the end of the path, forward motion stops.
8. To record your own flight path, see Terramodel Visualizer Help for instructions on using the Path toolbar.

Example of using the rotating mode

Open the mountain project as described in the first example.

1. Click NAVIGATION, and click Rotating Mode.
2. In the environment window, click and drag with the mouse. As you move the mouse right, left, up, and down, the project follows, always turning on its center.

If the project spins out of view, click NAVIGATION and Reset Camera. This is the position of the camera when it was created, and where this camera returns when you select Reset Camera. Each camera has its own reset position.

3. To smooth the display while moving, click the MOVING MODE and Move as Points.
4. Repeat step 2, rotating the project with the mouse. This time, whenever the project is in motion, the display changes to Points Mode, showing only the project points. Points Moving Mode provides the smoothest moving display performance, but select the Moving Mode you find easiest to use.
5. To move in closer to the project, press the Insert key. Press the Delete key to move out again. The Terramodel Visualizer Help lists other keyboard and mouse movement controls.

Example of using the flying mode

Open the mountain project as described in the first example.

1. Click NAVIGATION and Flying Mode.
2. Drag the mouse left to point the camera in that direction. The project seems to shift to the right. In Flying Mode, you are moving the camera, not the project, so the project moves opposite the mouse direction. If you release the mouse button while dragging, the camera will continue to fly on its own.

Note: Use the Reset Camera command to regain flight control.

3. To start forward flight, press the 1, 2, or 3 number key. The camera moves forward through the project.

Begin slowly - you can select a higher speed later. If the speed is still too high with the 1 key, Help has instructions for adjusting the velocity factor.

4. Steer the camera by dragging the mouse to point the camera in the direction you want to go. For example, if you are about to dive into the surface, drag the mouse up.

5. The right mouse button also controls flight steering. While continuing flight, click and drag the right mouse button to the left, then to the right. The view slips sideways without changing the forward orientation of the camera.
6. Click and drag the right mouse button up, then down. The right mouse button is another way to move in and out.

Example of saving an image

To record images of environments, Terramodel Visualizer includes a high-resolution screen capture utility. To create an example image, open the GOLF project as described in the first example.

Note: To use Flying and Rotating in GOLF, set the Moving Mode to Points first. GOLF is a complex project in Texture Mode and moves slowly on some machines.

1. To open the golf project environment, from the File menu, click Open. A dialog box appears asking to save the MOUNTAIN.EN file. Click No. The Open dialog box appears.
2. Open the GOLF folder containing the project and environment files.
3. Click GOLF.EN to open the environment file, and click OK.

The golf project appears with surface textures and images of trees. For more information about textures and other display options, see the Help.

4. Click FILE and Export Image. The SnapShot dialog box appears.
5. In the File Name text box, type a name for the screen capture. The extension is automatically entered.
6. For this example, in the Style Type box select Windows Bitmap *.BMP.
7. Use the default Image Properties settings as a start.
8. Click Save to create the image.
9. To display or print the Bitmap image, open the file in Microsoft Paint.

Explore other example projects

There are several example data sets that can help you learn how to use Terramodel Visualizer without first creating your own data.

GOLF.EN illustrates use of surface textures and cross-texture images for displaying landscaping projects.

METRO.EN file shows a road project added on to an existing street system. With the optional Orthodrape Module, you can also see how Terramodel Visualizer drapes orthographic images on the three-dimensional surface.

LANDFILL.EN illustrates displaying a complex three-dimensional construction.

Chapter Sixteen

How to Plot in Terramodel

Preparing to Plot from Terramodel

Windows-Supported Printing/Plotting Devices

Since **Terramodel** is a Windows application, you can print or plot to any of the printing devices supported by Windows. In the context of Windows, printers and plotters are called printers and are configured as a printer and controlled by the print manager. In the context of **Terramodel** the devices capable of producing large documents are called plotters. The terms printer and plotter and printing and plotting will somewhat be used interchangeably.

A supported printing device will have a printer-driver (a list of instructions that tells Windows about your printer and how to connect to it) either supplied with the Windows program, or supplied by the printer's manufacturer.

If your printer-driver is included in the list of drivers supplied with Windows, you can install it so that your printer will appear in the list of printers in the Windows *Print Manager* dialog box and in the list of Window's Printers in **Terramodel's** *Driver Selection* dialog box (**PlotterSet** command). See the Window's User's Guide or the

documentation provided with the printer/plotter, for information on how to install a printer-driver file.

If your printer-driver is not included with the Windows-supplied drivers, you will have to obtain it from your printer's manufacturer. In this case, your printer may or may not be supported by Windows. To determine if it is supported, you can check Windows' Hardware Compatibility List. Your printer may also have an emulation mode that will allow it to function as one of the printers supported by Windows. Check the Windows manual for other ways to make your printer/plotter Windows-compatible.

Please note that some printer/plotters do not support raster plots and hence can not plot a 3D-View or an image.

Direct Terramodel Plotter Drivers

Terramodel also supplies direct drivers for selected plotters. The list of supported plotters is continuously increasing as we develop new drivers for the emerging choices of plotters. The direct drivers do not have the same capabilities as the Windows printer drivers, the pros and cons of each driver type is discussed herein.

Configuring Your Plotter

Terramodel lets you define "plotter configurations", which let you store the plotting device, communications port, paper size, page layout, and pen carousel together under a single name. These named configurations are then listed in a dialog box, where you can select them to use for plotting, edit them, add new configurations, or delete old ones. Storing the plotting options under a single configuration saves time and keeps you from having to apply numerous settings each time you plot. Terramodel supports several key features that enable you to create reproducible plots from your design,

To select a plotter configuration:

- ▶ Enter **PlotterSet** at the command bar,
or
- Open the **FILE** menu and select **Plot Setup**, then **Plotter setup**.

The *Plotter Configurations* dialog box will open. The Configuration list box includes a list of plotter configurations that you have already established.

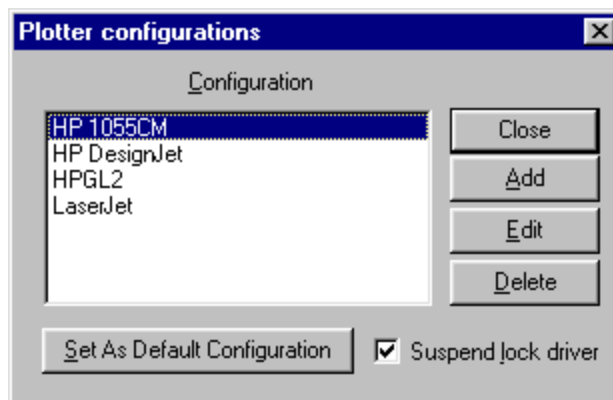


Figure 16-1 Plotter Configurations Dialog Box

- Select the plotter configuration of your choice.
- Click the **Set As Default Configuration** button to have Terramodel use the selected plotter configuration each time you plot.
- Click Close to exit the *Plotter Configurations* dialog box.

To add a plotter configuration:

- Enter **PlotterSet** at the command bar,
or
Open the **FILE** menu and select **Plot Setup**, then **Plotter setup**.

The *Plotter Configurations* dialog box will open.

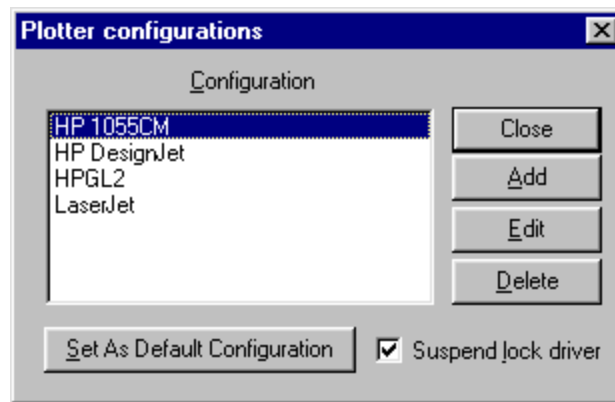


Figure 16-2 Plotter Configurations Dialog Box

- Click the **Add** button to open the *Add Plotter Configuration* dialog box.

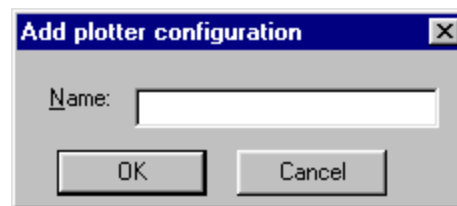


Figure 16-3 Add Plotter Configuration Dialog Box

- Enter a distinctive configuration name (up to 22 characters including spaces) that will enable you to recognize the plotter configuration from the Configuration list box (for example, *Designjet 24 X 36*). Click **OK** to advance to the *Plotter Configuration* dialog box.

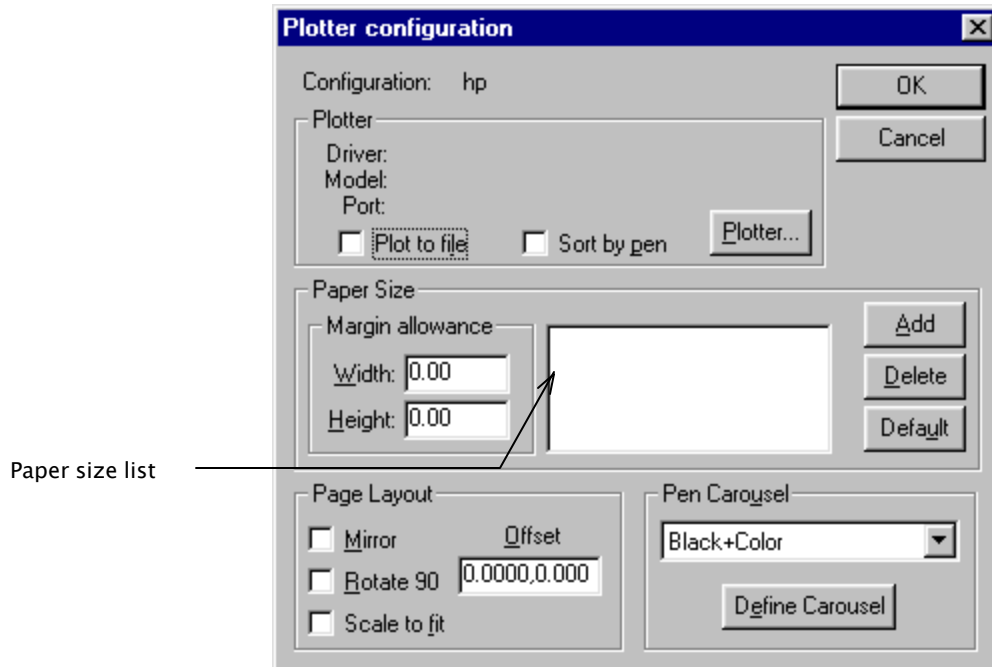


Figure 16-4 Plotter Configuration Dialog Box

The name of the plotter configuration you are storing is shown at the top of the dialog box. The information displayed in the Plotter section of the dialog box reflects the plotter driver, plotter model and communications port you selected. For new plotter configurations, this area will be blank until you select a plotter.

- Enter the total margin amount for the width and height of your plot in the Margin Allowance section. **Terramodel** subtracts these amounts from the total paper size to determine how much of the plotting paper can be used.
- In the Page Layout section, check the boxes if you want to mirror or rotate the plot, or have **Terramodel** scale the plot to fit within the paper limits. Enter a width, height offset in the Offset box if you want to offset your plot from the corner of the plotting paper.
- Click the **Plotter** button to open the *Driver Selection* dialog box to select a plotter driver and communications port.

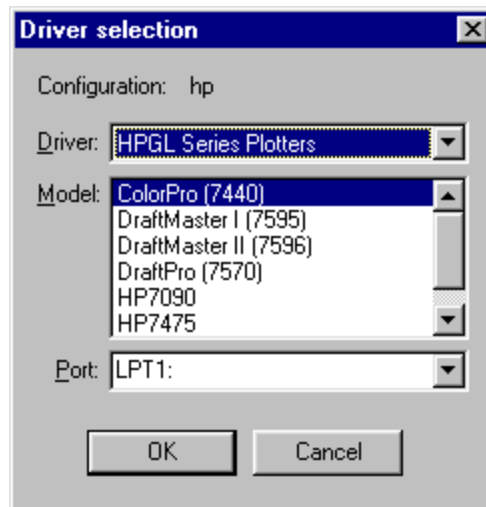


Figure 16-5 Driver Selection Dialog Box

- Click the down arrow next to the **Driver** drop-down list box to open it. Select a driver type. The types supported by are HPGL plotter series, HP-GL/2 series and Windows printers. The HPGL and HP-GL/2 series use direct drivers supplied by Spectra Precision Software, Inc. Note that only the Windows drivers support the plotting of raster images. Only the HPGL and HP-GL2 drivers support plotting of lengths over a few feet. Clients have used the direct drivers to plot sheets, such as long vertical alignments, over 100' in length (with the appropriate plotters).
- Use the scroll bar in the Model list box to select a plotter model. The driver type you selected in the Drivers list box controls the types of plotter models displayed in this list.
- Select a communications port from the list of ports available on your computer in the Ports list box. Click OK to exit the *Driver Selection* dialog box.
- In the *Plotter Configuration* dialog box, if you selected a driver other than the **Windows Printers** driver, you can select a paper size from defined paper sizes in the Paper Size list box. To define a paper size, see **To add a paper size** on page 565. If you selected the Windows Printers driver, **Terramodel** will take the paper sizes from the individual plotter/printer settings and will not show them here.
- In the Pen Carousel section, select a pen carousel for this plotter configuration from the drop-down list box. To define pen carousels, see **To define a pen carousel** on page 566.

- Click OK to save the plotter configuration.

To edit a plotter configuration:

- In the Configuration list box of the *Plotter Configurations* dialog box, highlight the configuration you want to change, then click the **Edit** button. The *Plotter Configuration* dialog box will open. Edit one or more properties in the configuration, then click OK to save the changes.

To delete a plotter configuration:

- In the *Plotter Configurations* dialog box, highlight the configuration you want to delete.
- Click the **Delete** button to remove the configuration from the list.

To add a paper size:

- In the *Plotter Configuration* dialog box, click the **Add** button in the Paper Size section. The *Custom Paper Size* dialog box will open.

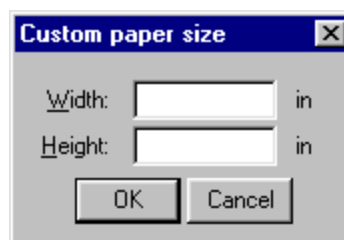


Figure 16-6 Custom Paper Size Dialog Box

- Enter the width and height, in inches, of the paper size you want to add in these real number controls. Click OK to close the dialog box. Your paper size will appear in the Paper Size list box of the *Plotter Configuration* dialog box.

To delete a paper size:

- Highlight the paper size that you want to delete in the list box.
- Click the **Delete** command button to remove the paper size from the list.

To set a default paper size:

- ▶ Highlight the paper size in the list box that you want **Terramodel** to use as the default size when you plot. Click the **Default** command button.

The **Default** button is dimmed if you selected the **Windows Printers** driver in the *Driver Selection* dialog box.

Using Pen Carousels

Through the use of pen carousels, **Terramodel** provides you with a way to define the colors and line widths that will be drawn with plotter pens, as well as the pen speeds of pen plotters. You can define multiple pen carousels, then select the one you want to use from a list of pen carousels.

To define a pen carousel:

- ▶ Click the **Define Carousel** button in the *Plotter Configuration* dialog box,

or,

Open the **FILE** menu and select **Plot Setup**, then **Carousel setup**.

The *Edit Pen Carousel* dialog box will open.

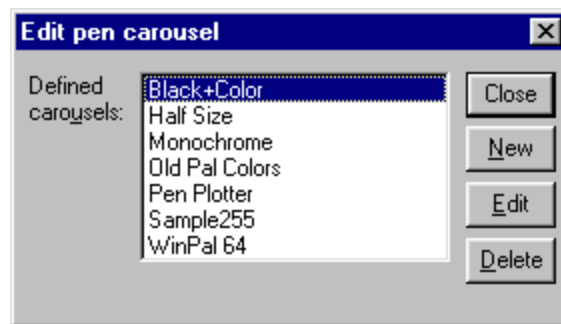


Figure 16-7 Edit Pen carousel Dialog Box

- ▶ Click the **New** button to open the *Add Pen Carousel* dialog box.

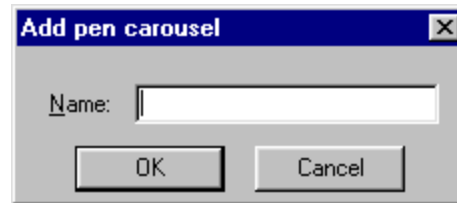


Figure 16-8 Add Pen carousel Dialog Box

- Enter up to 22 characters, including spaces, in the **Name** text box for a description of the pen carousel. Click **OK** to advance to the *Pen Carousel Definition* dialog box.

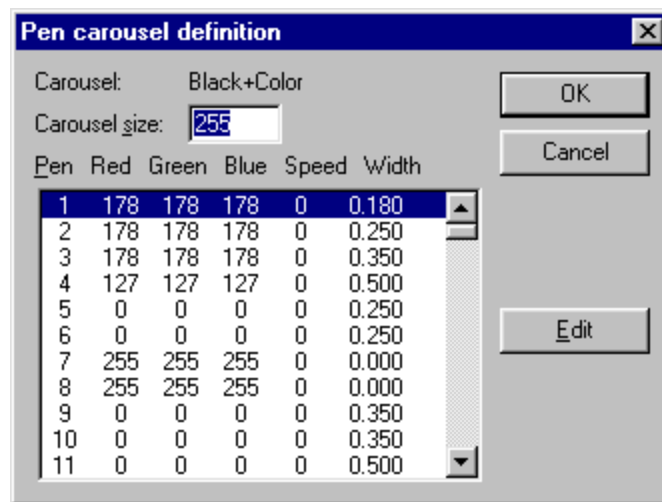


Figure 16-9 Pen Carousel Definition Dialog Box

- Enter the number of pens you want to define (up to 255) in the **Carousel Size** box. Highlight the line of the first pen you want to define, then click the **Edit** button to open the *Pen Definition* dialog box.

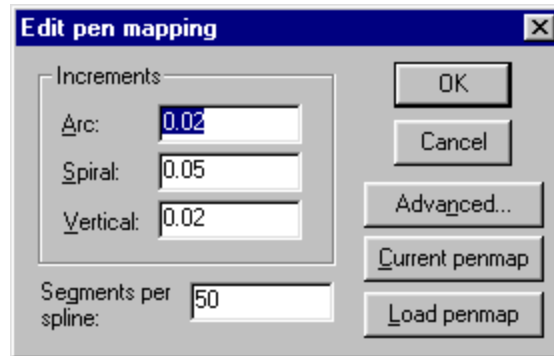


Figure 16-10 Pen Definition Dialog Box

- For raster plotting devices, enter the contribution amounts (from 0 to 255) of red, green and blue for the color that the pen will plot in the corresponding boxes in the Color section. You can use the color contribution amounts of a particular color by clicking the **Choose** button to open Window's *Color* dialog box and selecting a color box from that dialog box.
- For raster plotting devices, enter the width of the lines (in millimeters) that will be plotted by this pen.
- For pen plotters, enter the velocity of the pen in the Pen Speed box. Entering a velocity of zero (0) causes **Terramodel** to use the plotter's default speed for that pen number.
- Click OK to return to the *Pen Carousel Definition* dialog box. Continue defining the remaining pens in the pen carousel definition in the same manner. When you have finished, click OK to return to the *Edit Pen Carousel* dialog box. Your new pen carousel definition will appear in the list of defined carousels.

To edit a pen carousel definition:

- Highlight the pen carousel you want to change in the Defined Carousels list box. Click the **Edit** button to open the *Pen Carousel Definition* dialog box.
- Highlight the line of the pen number you want to change. Click the **Edit** button to open the *Pen Definition* dialog box.
- Edit the red, green or blue contribution amounts, or change the width or pen speed. Click OK to save the changes. Click OK in the *Pen Carousel Definition* dialog box to close it.

To delete a pen carousel definition:

- ▶ Highlight the pen carousel definition you want to delete in the Defined Carousels list box.
- ▶ Click the **Delete** button to remove the definition from the list.

Setting Plot Resolutions

The **PlotSet** command (File menu, Plot setup sub-menu, Plot params option) opens the *Edit Pen Mapping* dialog box, where you can set the smoothness of plotted curves.

To set the curve smoothness:

- ▶ Enter **plotset** at the command bar,
or
Open the **FILE** menu and select **Plot setup**, then **Plot params**. The *Edit Pen Mapping* dialog box will open.

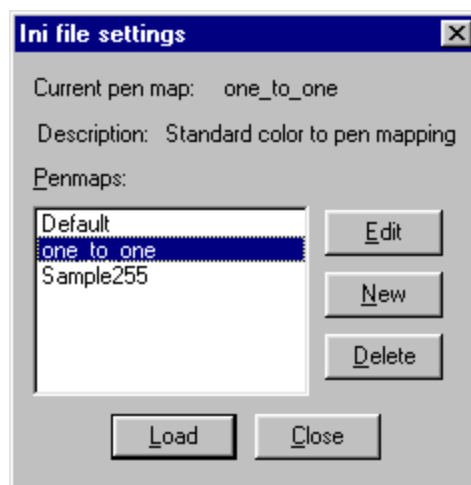


Figure 16-11 Edit Pen Mapping Dialog Box

- ▶ **Terramodel** actually plots curves as very short, straight-line segments. The length for arcs, spiral curves, or vertical curves are defined independently. Enter the length of each straight-line segment in sheet units as selected in the **MeasUnit** command. Generally, a value of 0.05 inches provides reasonably smooth curves. **Terramodel** plots a minimum of 4 straight-line segments per arc.

- Enter the smoothing factor (between 0 and 100) for splined polylines, such as contour lines, in the Segments Per Spline box. This value is roughly equivalent to the number of points placed on a contour curve (spline) between triangle links. Entering 0 is the same as plotting unsplined lines. A value of 10 usually provides adequately smooth curves. Factors above 10 don't dramatically increase the smoothing, and in fact increase the plotting time since contours are calculated as they are plotted.
- Click OK to save the settings.

Mapping Colors to Pens

The **PlotSet** command opens the *Edit Pen Mapping* dialog box, where you can also map colors to plotter pens. **Terramodel** lets you store each pen mapping definition under a separate name, which you can select from a list of stored pen maps.

To load a pen mapping definition:

- Enter **plotset** at the command bar,
or
Open the **FILE** menu and select **Plot setup**, then **Plot params**. The *Edit Pen Mapping* dialog box will open.
- Click the **Load penmap** button to open the *Ini File Settings* dialog box.

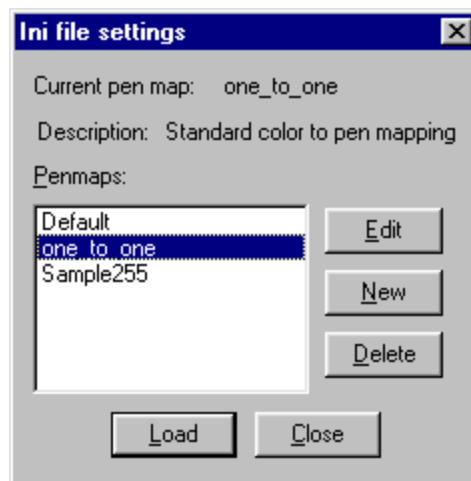


Figure 16-12 Ini File Settings Dialog Box

- Highlight the name of the pen map definition you want to load in the Penmaps: list box. Click the **Load** button. Terramodel will use the selected pen mapping configuration as the default whenever you plot.
- Click **Close** to close the *Ini File Settings* dialog box.

To create a new pen mapping definition:

- Enter **plotset** at the command bar,
or
Open the **FILE** menu and select **Plot setup**, then **Plot params**. The *Edit Pen Mapping* dialog box will open.
- Click the **Load penmap** button to open the *Ini File Settings* dialog box.
- **Terramodel** supplies a default pen mapping definition called **Default**, which is included in the list and can not be edited. Click the **New** button to open the *New Penmap* dialog box.

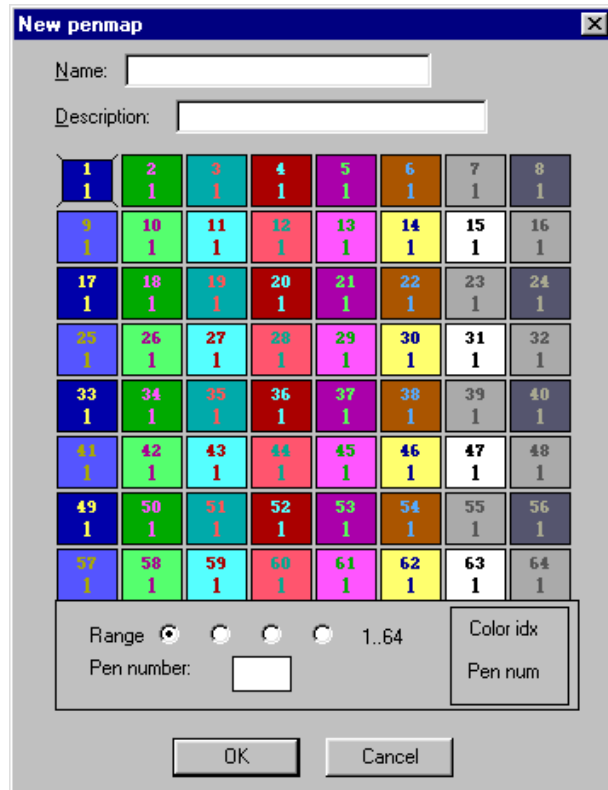


Figure 16-13 New Penmap Dialog Box

- Enter up to 22 characters in the Name edit text box to assign a name that will let you identify the pen mapping configuration from the list in the *Ini File Settings* dialog box.
- Enter a brief description (also maximum 22 characters) of the pen mapping in the Description edit text box.
- In each color box, the color numbers are displayed at the top. The number of the pen assigned to the color is immediately below each color number. To change a pen number, click the box you want to edit. Type the pen number you want to assign to that color in the **Pen number:** box at the bottom of the dialog box.
- You can view different ranges of color numbers and their corresponding pen numbers by clicking the Range radio buttons. Each radio button displays a group of 64 color numbers. Change the pen numbers assigned to the different ranges of colors in the same manner as described above.

- Click OK to save the pen mapping configuration. The name of the definition will appear in the Penmaps: list box in the *Ini File Settings* dialog box.

To edit the current pen mapping definition:

- Enter **plotset** at the command bar,
or
Open the **FILE** menu and select **Plot setup**, then **Plot params**. The *Edit Pen Mapping* dialog box will open.
- Click the **Current penmap** button to open the *Edit Current Penmap Settings* dialog box.

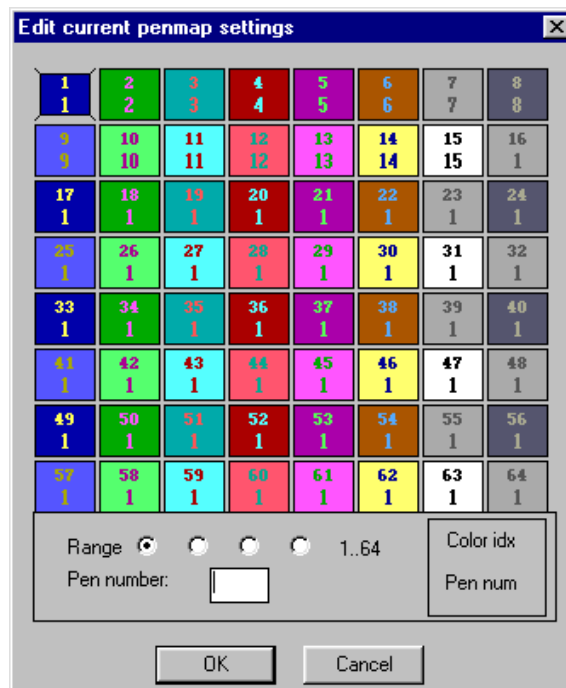


Figure 16-14 Edit Current Penmap Settings Dialog Box

- In each color box, the color numbers are displayed at the top. The number of the pen assigned to the color is immediately below each color number. To change a pen number, click the box you want to edit. Type the pen number you want to assign to that color in the **Pen number:** box at the bottom of the dialog box.
- You can view different ranges of color numbers and their corresponding pen numbers by clicking the Range radio buttons. Each radio button displays a group of 64 color

numbers. Change the pen numbers assigned to the different ranges of colors in the same manner as described above.

- Click OK to save the pen mapping configuration.

To edit a pen mapping definition other than the current one:

- Enter **plotset** at the command bar,
or
Open the **FILE** menu and select **Plot setup**, then **Plot params**. The *Edit Pen Mapping* dialog box will open.
- Click the **Load penmap** button to open the *Ini File Settings* dialog box.
- In the Penmaps: list box, highlight the name of the pen map definition you want to change. Click the **Edit** button to open the *Edit Pen Map Selection* dialog box.

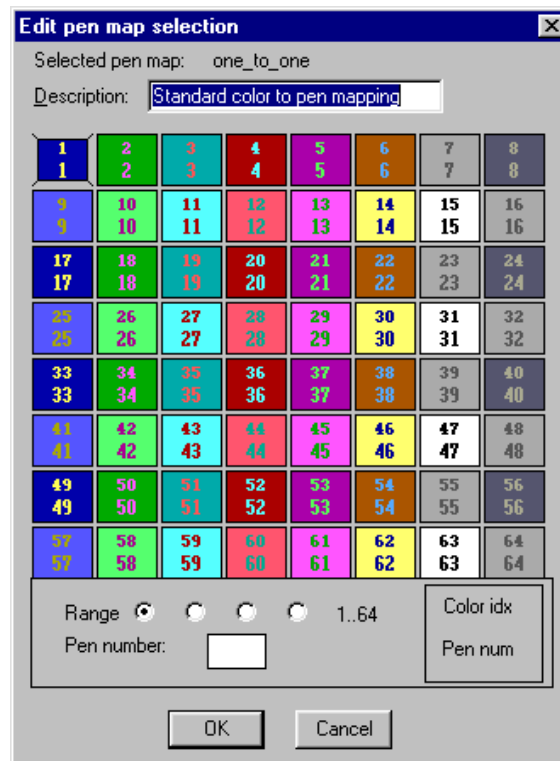


Figure 16-15 Edit Pen Map Selection Dialog Box

- Enter up to 22 characters for a new description of the pen map definition in the Description edit text box.
- In each color box, the color numbers are displayed at the top. The number of the pen assigned to the color is immediately below each color number. To change a pen number, click the box you want to edit. Type the pen number you want to assign to that color in the **Pen number:** box at the bottom of the dialog box.
- You can view different ranges of color numbers and their corresponding pen numbers by clicking the Range radio buttons. Each radio button displays a group of 64 color numbers. Change the pen numbers assigned to the different ranges of colors in the same manner as described above.
- Click OK to save the pen mapping configuration.

Advanced plotting features

Terramodel supports several features that enable the user to plot all of the sheets of a project without having to change plotting configurations or any of the objects. The *advanced* options are used to enable the color/pen shift when using layer lists and background prominence settings, and pen mapping based on text height.

The use of layer lists is described in chapter 5 of this guide. When the optional background prominence layer settings are used with layer list applied to a dynaview, objects on the selected layers will be plotted and or displayed using the the specified color and pen shift. This is done so that in one dynaview objects on selected layers can appear solid and in another dynaview they can appear screened (requires a raster plotter).

When using this feature the pen carousel should be created so that when the shift is applied the pens that will be used will be screened. For example pens 1-16 could be solid and 17-32 could be screened. When a shift of 16 is applied objects with a color of one will be plotted with the pen that would be used to plot color 17. If you want to also see on the display a color shift you may also enable the color shift option.

In addition to plotting objects based on the color of the object **Terramodel** has the unique ability to select the pen to be used when plotting text objects based on the height of text. When this feature is used the proper pen is always selected based on the height of the text. Normally smaller text requires a smaller pen and larger text a larger pen. The chart shown in the *Advanced pen map settings* dialog box below shows how the four size ranges that can be used. For each range you may specify the pen to be used. If the height of a text object does not fall within the text ranges specified the pen used will be based on the color of the text.

To set the advanced pen mapping settings:

- ▶ Enter **plotset** at the command bar, the *Edit Pen Mapping* dialog box will open.
- ▶ Click the **Advanced** button to open the *Advanced pen map settings* dialog box.

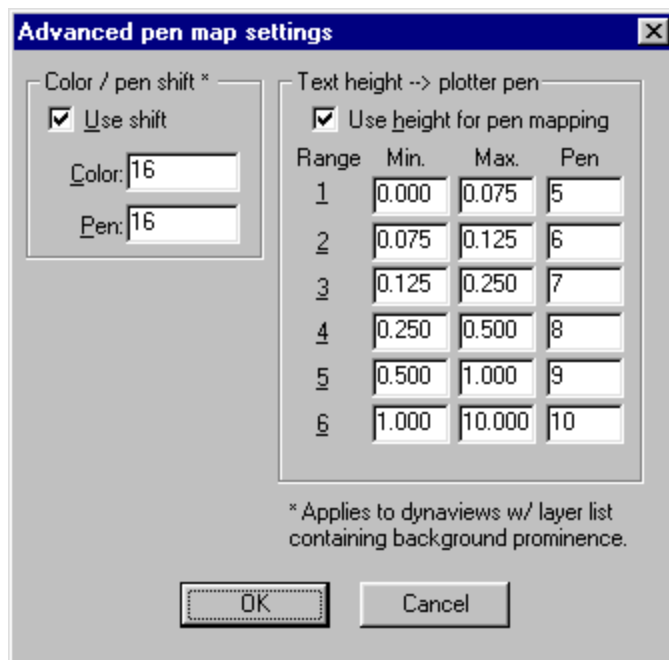


Figure 16-16 Advanced Pen Map Settings Dialog Box

- ▶ You may optionally enable a color/pen shift and pen mapping based on text height.
- ▶ Click **OK** to close the *Advanced pen map settings* dialog box.

Plotting A Project File

When you plot from **Terramodel**, you can choose to plot an area in your project file defined by window corners, or you can plot the objects displayed within the limits of a box you created using the **Plotbox** or **Box** command. You can store and plot these boxes from any view.

After you've drawn your plotboxes, you can organize them into plot sheets in the sheet view using dynaviews. If you have the ASAP module this is done in a highly automated fashion for most projects. In some cases, such as in combination plan-profile sheets, you'll have several dynaviews contained in a single plot sheet. If you place a box around the group of dynaviews, you can select that box when you use the Box option of **Plot**. For more information on creating dynaviews, see chapter 4, **Understanding Terramodel Objects**, "Understanding Dynaviews", and chapter 5, **Understanding Terramodel CAD**, "Working With Dynaviews".

Plotting to a Device or a File

You can plot directly to a Windows-supported printer/plotter device, or have **Terramodel** store the pen movements for the plot in a file.

To plot to a printer or plotter:

- Enter **plot** at the command bar,
or
Open the **FILE** menu and select **Plot**.
- When the **Plot** command bar opens, the last plot option will be the default and the focus will be on the entry of the information required for plotting. You may choose to plot by Window and the default corners will be the extents of the active view mode. You may also plot by box and select a plot box to define the limits or you may choose to plot by selecting multiple plot boxes. The multiple option enables you to plot all of the sheets generated by the ASAP module in one operation or all of the hydrographs generated by the HYDRO commands.
- Define your plotting area, then click OK to open the *Plot* dialog box.

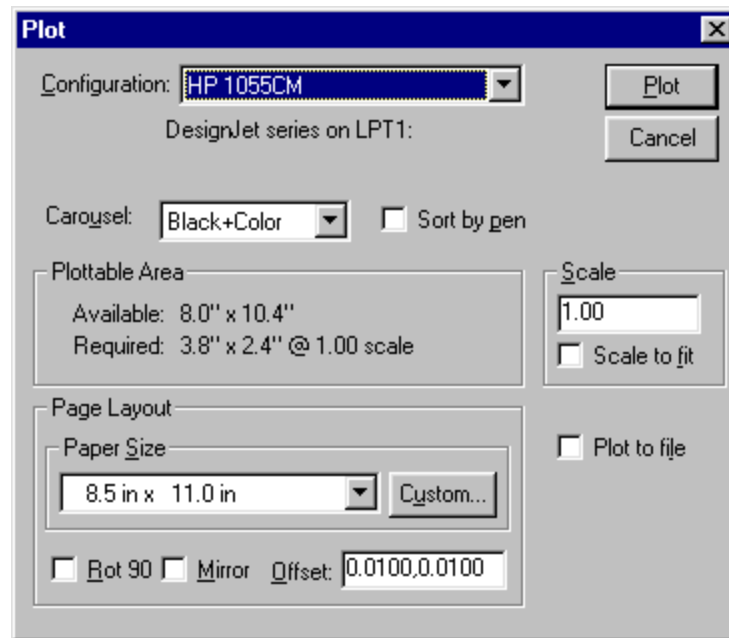


Figure 16-17 Plot Dialog Box

- The plotter configuration includes the plotter driver **Terramodel** is currently using and the communications port. The name of the current configuration is shown in the Configuration list box. To change the plotter configuration, click the down arrow to open the list box and select a different plotter configuration. (To store a plotter configuration, see page 561.) When you change the plotter configuration, **Terramodel** will display the new configuration's plotter driver and port immediately below the Configuration list box.
- To choose a different pen carousel definition, click the down arrow next to the **Carousel:** drop down list box and select a new definition.
- If you are plotting to a pen plotter, you can check the **Sort by pen** check box to have **Terramodel** sort and plot objects by color. The pen for color number one is determined and all colors that use this pen will be plotted. This process continues working from the lowest color number to the highest. For Windows printers this is the only control the user has as to which pen will be plotted on top of other pens. When plotting using the Windows printer it is recommended that pens of lighter colors (shaded) be assigned (and hence plotted) to the lowest **Terramodel** color numbers. This method is not required when using the direct plotter driver for HP-GL/2 plotters as the driver issues a command to the plotter to put darker lines on top.

- **Terramodel** displays the available and required plotting areas for this plot based on the paper size, margin allowance, plotbox or window size, and plot scale. As you change any of those parameters, **Terramodel** automatically updates this area.

If you selected the **Windows Printers** driver from the *Driver Selection* dialog box of **PlotterSet**, clicking the **Setup** button will open the *Setup* dialog box for the corresponding printer/plotter device, where you can establish paper size, orientation, and other plotting features specific to that device. The **Setup** button does not appear in the *Plot* dialog box if you selected one of the other driver types.

- To set the plot scale, enter the scale (in feet per inch or a ratio when using meters) at which you want **Terramodel** to plot your drawing in the box in the Scale section. If you want **Terramodel** to scale the plot to fit within the available plotting area, check the **Scale to fit** check box.

- To choose a paper size from the list of paper sizes you defined in the *Plotter Configuration* dialog box (**PlotterSet** command), click the down arrow next to the Paper Size drop-down list box to open it. Select a paper size. If you selected the **Windows Printers** driver from the *Driver Selection* dialog box of **PlotterSet**, the available paper sizes are determined by each printer/plotter's Windows setup and are not shown in this list.

If your desired paper size is not included in the list (and you are not using the **Windows Printers** driver), click the **Custom** button to open the *Custom Paper Size* dialog box, where you can enter a paper width and height. The **Custom** button is dimmed when you select the **Windows Printers** driver from the *Driver Selection* dialog box of **PlotterSet**.

- If you want to rotate your plot by 90 degrees or plot a mirror image of the plotting area, check the appropriate box. If you want to offset your plot from the lower left corner of the plotter's plotting limits, enter the width, height offset values in the Offset box.



- To plot directly to your plotting device, **do not** check the **Plot to File** check box.
- Click **Plot** to plot the project.

To plot to a file:

- ▶ Enter **plot** at the command bar,
or
Open the **FILE** menu and select **Plot**.
- ▶ Follow the steps outlined in **To plot to a printer or plotter**, but check the **Plot to File** check box. The *Plot File* dialog box will open.



Figure 16-18 Plot File Dialog Box

- ▶ In the **File Name** text box, type the file name, with extension, that will contain the plotting information. If you want to store the file in a different directory, click the appropriate file folder icon in the **Directories** list box. If you want to store the file on another drive, click the down arrow next to the **Drives** drop-down list box to open it and select another drive.
- ▶ Click **OK** to send the plot to a file.

Chapter Seventeen

Digitizer Setup

This chapter explains how to set up communication between Terramodel and your digitizer, define menus to select Terramodel commands from the digitizer, and define a screen area in which you can use your digitizer as a mouse. A digitizer enables you to trace in information from paper to Terramodel using the scale and orientation of the map.

Terramodel supports the use of a mouse and a digitizer at the same time.

Get Your Digitizer Ready

Before you can begin using your digitizer, do the following:

- Be sure all digitizer connections are correctly attached to your computer according to documentation furnished with your digitizer.
- Make sure the digitizer is plugged in and turned on.
- Attach to the tablet templates for any tablet menus you want to use. A template can be plotted from the template.pro project file.
- Mark off any screen areas you want to use on the tablet. Within a screen area, the digitizer behaves like a mouse. You can use it to move the pointer to any place on the screen and to select menus and buttons in Terramodel or other applications.

- Attach the plan sheet or map you want to digitize. Be sure that the document does not overlap tablet areas you have designated as menus or screen areas.

Configuring Your Digitizer

Supported digitizer tablets and their specific settings are listed at the end of this chapter.

To use your digitizer with Terramodel, you must turn on the digitizer from the Tablet command bar at the start of each digitizing session.

To configure your digitizer in Terramodel:

1. Start Terramodel, and open a new or existing project file.
2. Type **Tablet** into the command bar.
OR
Click the FILE menu, **System configuration**, and **Tablet**.

The Tablet command bar opens near the bottom of the screen.



3. Click **Setup**. The *Digitizer Setup* dialog box appears.



4. From the Button Mapping File list, select **TMODWIN.BTN**.
The list displays all button mapping files (*.BTN) located in the Terramodel folder.

The Button Mapping File defines the Terramodel commands assigned to the digitizer buttons. To add or change button assignments, see Terramodel Help.
5. Select your digitizer in the Digitizer Model list,.
6. Select the communication settings for your digitizer.
Refer to the tablet settings later in this chapter and your digitizer documentation for the correct entries.
7. Click **OK** to close the dialog box.
8. Click the **Off** button on the Tablet command bar. The button text changes to On, and the Orient Menu and Orient Map buttons are made available.

The OFF/ON button shows the status of the digitizer. When the digitizer is on, the button displays On. When the digitizer is off, the button displays Off. The digitizer must be turned on in Terramodel before you can orient menus and maps or enter project data.

Orienting Digitizer Menus and Screen Areas

For improved productivity, set up optional menus on your digitizer from which you can select Terramodel commands. Menus enable you to do virtually all of your Terramodel

work from the digitizer, rather than switching back and forth between the digitizer, the mouse, and the keyboard.

Designate an optional screen area to expand digitizer control to the entire screen. Within the screen area you can use the digitizer instead of the mouse to select Terramodel or Windows buttons and menu commands—even start and control other applications.

Menu and screen area orientation settings are saved to the currently selected menu file. These settings are used for subsequent digitizer sessions, until you select a different menu file or change the orientation of a menu or screen area.

Default Menus and Screen Area

The *Digitizer Menu Orientation* dialog box lists two predefined menus and a screen area. The TEMPLATE.PRO file installed with Terramodel contains a template for the menu. Plot the templates at a scale you find convenient and attach it to the tablet. The screen area can also be any size.

Each menu and screen area is oriented separately, so you can place them anywhere on the tablet: together in a group, in a row along a tablet edge, or in different corners.

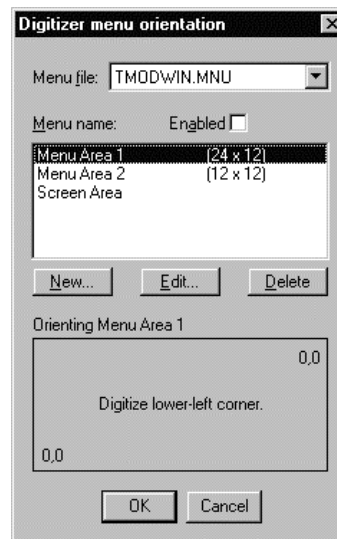
You can define up to 16 menus and/or screen areas, five of which can be active at the same time. See “Creating a New Menu” and “Creating a New Screen Area” in this chapter for instructions on making your own menus.

Orienting Menus and Screen Areas

Orient and enable menus and screen areas in the same manner. The location of each is defined by digitizing the lower-left and upper-right corners of the menu on the tablet.

To orient a menu or screen area on the tablet:

1. Click the **Off** button on the Tablet command bar to turn on the digitizer.
2. Click **Orient Menu** to display the *Digitizer Menu Orientation* dialog box.



3. Select a file from the Menu File list. The default selection is TMODWIN.MNU. The Menu File box shows a list of all the menu files found in the Terramodel Search Path, from the current project folder to the Terramodel installation folder.
4. Click the menu you want to orient on the Menu Name list. The text area at the bottom of the dialog box changes to Orienting Menu Name and Digitize Lower-Left Corner. If you have not oriented the menu before, the corner coordinates are 00,00 and 00,00.
5. Digitize the lower-left and upper-right corners of the menu, following the prompts on the dialog box. As you enter them, tablet coordinates for each corner appears on the dialog box. After you enter both corners for the selected item, the selection moves to the next item on the Menu Name list.

To enable a menu or screen area for use:

1. Click the menu or screen area name in the Menu Name list.
2. Click the **Enabled** check box.
An E appears on the Menu Name list following the selected name.
3. Click **OK** to close the dialog box.

To deactivate a menu or screen area:

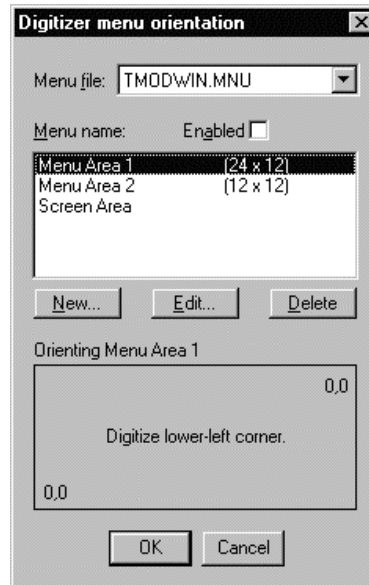
1. Select the enabled menu or screen area name from the Menu Name list.
2. Click the **Enabled** check box.
The E following the selected name is removed from the Menu Name list.
3. Click **OK** to close the dialog box.

Creating a New Menu

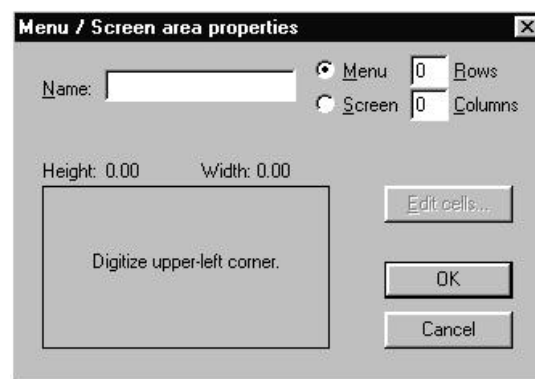
Although Terramodel includes menu templates with the most commonly used commands and keys, you can create your own menu layouts using the Menu/Screen Area Properties and Edit Menu Cell dialog boxes. For example, to accommodate a large map area on the tablet, you can create menus as strips along a tablet edge. You can also create menus that include just the commands you need for each phase of a project.

To create a new tablet menu:

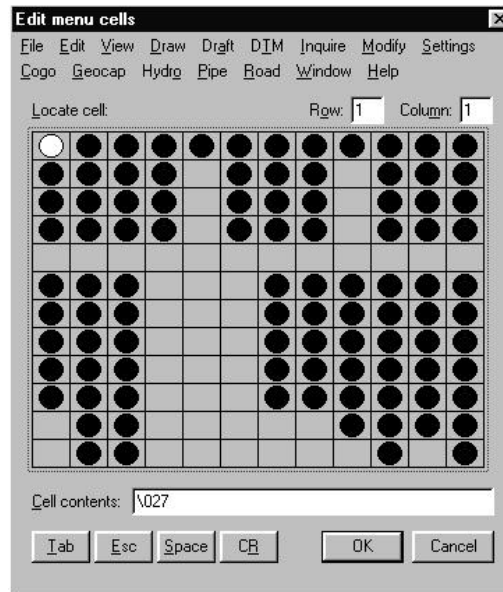
1. Affix the template for the new menu to the tablet.
2. Click **Orient Menu** on the Tablet command bar. The *Digitizer Menu Orientation* dialog box appears.



3. Click **New**. The *Menu/Screen Area Properties* dialog box appears.



4. In the **Name** text box, type in an identifying name, for example: Menu Area 3.
5. Type the number of rows and columns you want on the menu.
6. Set the height and width of the menu area by digitizing the upper-left, lower-left, and lower right corners of the template, as prompted on the dialog box. Asterisks marking entered coordinates and the calculated height and width appear on the dialog box.
7. Click **Edit Cells** to display the *Edit Menu Cells* dialog box.



While the Edit Menu Cells dialog box is displayed, the screen pointer cannot be controlled by the digitizer in a screen area. Use the mouse or keyboard to make changes.

8. Click the cell you want to change on the **Locate Cell** matrix. Cells assigned commands are marked with a black dot on the Locate Cell matrix. The currently selected cell is marked with a white dot. The contents of the selected cell appears in the Cell Contents text box.
9. With the mouse, select the command you want from the menus at the top of the dialog box. The new command replaces the previous contents in the Cell Contents text box.
OR
Type the command name in the **Cell Contents** text box.
10. Click the appropriate button at the bottom of the dialog box to include an Escape, Tab, Space, or Carriage Return (Enter) key stroke with a command. Many Terramodel commands require a carriage return to activate.
11. To clear the contents of a cell, select the text in the Cell Contents box and press Back Space to delete it. The dot for that cell is removed from the cell map.
12. After you assign commands to menu cells, click **OK** to close the *Edit Menu Cells* dialog box.
13. Click **OK** to close the *Menu/Screen Area Properties* dialog box.

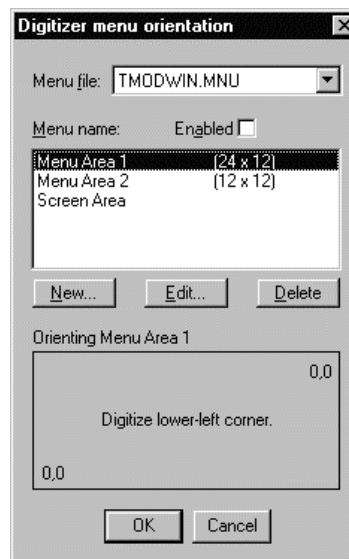
14. Orient and enable the new menu, following the steps in the previous section.
15. Click **OK** to close the *Digitizer Menu Orientation* dialog box.

Creating a New Screen Area

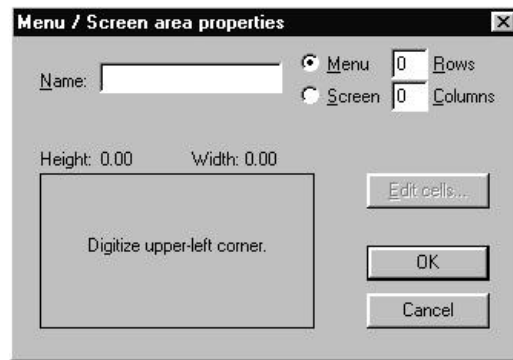
To access the screen more conveniently on larger digitizer boards, you can create one or more additional screen areas.

To create a new screen area:

1. Mark the new screen area on the tablet.
2. Click **Orient Menu** on the Tablet command bar. The *Digitizer Menu Orientation* dialog box appears.



3. Click **New**. The *Menu/Screen Area Properties* dialog box appears.



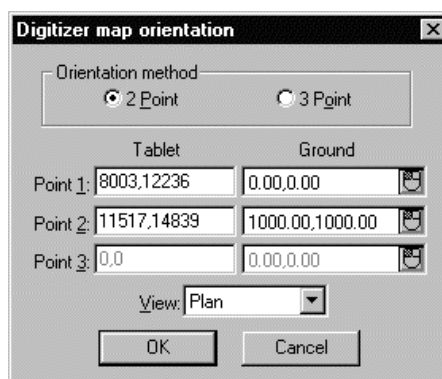
4. In the **Name** text box, type in an identifying name, for example: Screen Area 2.
5. Click **Screen Area**.
The rows and columns text boxes are unavailable and appear dimmed.
6. Set the height and width of the screen area by digitizing the upper-left, lower-left, and lower-right corners of the area on the tablet.
Asterisks appear on the dialog box as you digitize each corner. The calculated height and width also appears on the dialog box.
7. Click **OK** to close the *Menu/Screen Area Properties* dialog box.
8. Orient and enable the new screen area, following the instructions above.
9. Click **OK** to close the *Digitizer Menu Orientation* dialog box.


Orienting the Digitizer to Your Project

To use the digitizer to locate points on a plan sheet or map, you must orient the tablet to the coordinates of your project. You need to do this whenever you start a new project or begin a new session on an existing project.


To orient the plan sheet or map:

1. Affix the plan sheet securely to the digitizer
2. Click **Orient Map** on the Tablet command bar.
The *Digitizer Map Orientation* dialog box appears.



3. Click the **Orientation Method** you want to use.
The default selection is **2Point**, indicating the X and Y axes have the same scale. Use the **3Point** method when the X and Y axes have different scales. For example, when tracing in cross sections or a vertical profile, the horizontal and vertical axis typically use different scales.
4. Click the **View** list, and click the project view in which you want to enter data from the digitizer.
The map area is active only when the view selected on the dialog box is active.
5. Click the Point 1 Tablet box.
6. Digitize the **lower-left corner** of the map area on the tablet.
The selection moves to the Point 1 Ground text box.
7. Click  to enter the coordinates from the screen.
OR



Use the point control to enter northing and easting values in the Point 1 Ground box.

8. Click the Point 2 **Tablet** text box, and digitize the **upper-right corner** of the map area on the tablet.
The selection moves to the Point 2 Ground box.
9. Click  to enter the coordinates from the screen.
OR
Use the point (coordinate) control to enter northing and easting values in the Point 2 Ground box.
10. If you selected the 3Point method, repeat step 7 in the Point 3 Ground box for the third point.
11. Click **OK** to close the dialog box.

Entering Ground Coordinates from the Screen

You can enter the Digitizer Map Orientation Ground northing and easting coordinates directly from the screen. If you have oriented a screen area, use the digitizer to control the pointer, or you can use a mouse to follow the same steps.

To enter coordinates with the digitizer cursor:

1. With the Point 1 Ground box selected, move the digitizer within the **screen area** on the tablet and place the pointer on the dialog box.
2. Click . The dialog box is hidden from view.
3. Click the **lower left corner** of the project on the screen.
The Digitizer Map Orientation dialog box displays again, with the northing and easting values entered in the Point 1 Ground box.
4. Click  for the next the box.
5. Click the **upper right corner** of the project area on the screen.
The *Digitizer Map Orientation* dialog box displays again, with the northing and easting values entered for Point 2 Ground.
6. After you enter all necessary information on the Digitizer Map Orientation dialog box, click **OK**.

WinTab tablet driver

Most tablets are now supplied with a WinTAB Windows tablet driver. If you did not receive a WinTAB driver with your tablet contact the manufacturer to see if one is now available.

Select the following tablet:

From the **Tablet** command, setup option, select WinTAB.

Each tablet manufacturer writes a specific driver for each tablet series. This driver is only available from the manufacturer and must be installed before starting **Terramodel**. When using the WinTAB driver you may use the tablet to control other Windows applications just like a mouse. Most drivers enable you to use the mouse or tablet. When the WinTAB driver is specified within **Terramodel** the Tablet command has one more option, *Mouse swap*. This option enables you to switch between using the tablet as a tablet and mouse. This swap can be done from a toolbox or special function key by using the “**Tablet mo**” command.

When using the WinTAB driver it may be necessary to select the menu and orient options of the **Tablet** command from the keyboard.

Chapter Eighteen

Customizing Terramodel

This chapter contains a discussion of the tools you can use to customize **Terramodel** to suit your needs. In it you will find out how to organize commands you use most often into toolboxes that you can open and close when needed. You can also create custom icons and map them to buttons in the toolboxes, so that you can readily identify the commands you have organized.

You will also learn how to map commands to buttons on the Toolbar, how to map commands to the special function keys on your keyboard (f1–f12), and how to use the **Terramodel** Macro Language to create you own commands.

Using the ToolBox

ToolBox is an easy-to-use, easy-to-customize interface for **Terramodel** for Windows. With ToolBox you can:

- Group commands into toolboxes, which you can open, move, reshape, minimize, and close according to your drawing needs. You can open as many toolboxes as you like.
- Create your own toolboxes. ToolBox's multiple toolboxes provide a convenient way to access and sort your design and drafting tools.
- Create personalized workspaces. Save several ToolBox workspaces for various design, drafting and editing tasks.
- Create your own buttons. Customize your tool set on the fly. Assign two commands to each button (both left and right mouse buttons), and edit the pop-up notes text for each button.
- Create cascading toolboxes which can put hundreds of drawing tools at your fingertips without cluttering your screen.
- Drag and drop to move, copy, or remove command buttons from a toolbox.
- Drag and drop to link a toolbox to a cascade button in another toolbox.
- Tear off a toolbox from a cascade button.
- Design pictures for your custom commands with ToolBox's Bitmap Editor.
- Use your existing **Terramodel** menu.

Getting Started

ToolBox was automatically installed when you installed **Terramodel**.

ToolBox can be started from the command line by entering **ToolBox** or from the Toolbar by double-clicking the ToolBox icon. When started, ToolBox automatically attaches itself to your **Terramodel** session.

When you start ToolBox, a *workspace* opens that includes several *toolboxes*. Each toolbox is a collection of *command buttons*.

Using Workspaces

A *workspace* is a group of toolboxes saved in specific sizes and screen positions. You can open, close, move, reshape, minimize, import, delete, and create new toolboxes to create a custom working environment. This is your workspace.

A workspace remembers which toolboxes are open, the location of each toolbox, the size of each toolbox, and which toolboxes are minimized. You can save multiple workspaces for different design, drafting and editing tasks.

A workspace saves this information:

- Toolbox definitions.
- Open and closed toolboxes.
- Location of each toolbox.
- Shape of each toolbox.
- Minimized or normal display for each toolbox.
- Command buttons in each toolbox.
- Bitmap icons in use in the workspace.

The Default Workspace

When you start ToolBox for the first time, the default workspace loads. After the first time, ToolBox loads the last workspace you used. When you exit ToolBox, ToolBox remembers the current workspace name as the startup workspace for the next session.

Opening a Workspace

You can open any saved workspace to use with your drawing. When you open a workspace, each toolbox displays in the size and location in which it was last saved.

To open a workspace:

- From the Toolbox control menu on any toolbox (opened by clicking the Toolbox Menu Button in the upper-left corner of the toolbox), choose **Open Workspace**. The *Open Workspace* dialog box appears.

- Select name of the workspace you want to open.
- Click OK. The old workspace closes and the new one appears on screen.

If you have changed your workspace, you will have the opportunity to save it before the new workspace opens.

Creating a New Workspace

You can save several different toolbox setups by saving each workspace with a different name.

To create a new workspace:

- Arrange your toolboxes on the screen.
- From the Toolbox control menu on any toolbox, choose **Save Workspace As**. The *Save Workspace As* dialog box appears.
- Enter a name for the new workspace. The default name is the name of your current workspace. If you do not enter an extension, ToolBox adds the extension *.WS* to the new workspace file.
- Choose the OK button.

Saving a Workspace

Until you save a workspace or exit with the Save Workspace on Exit option turned on, ToolBox does not save the changes you make to the on-screen display of toolboxes.

To save your workspace:

- Open the Toolbox control menu on any toolbox.
- Select **Save Workspace**. ToolBox saves the current workspace.



If Save Workspace on Exit is checked, ToolBox automatically saves your workspace when you exit Terramodel or ToolBox. You can turn this feature off if you wish.

To turn off the automatic save feature:

- ▶ Open the Toolbox control menu on any toolbox.
- ▶ Select **Save Workspace on Exit** to remove the check mark.

The Library Workspace

The Library workspace contains command buttons and bitmap icons you can use to create custom toolboxes. It also contains all the toolboxes found in the default workspace. You can use these to replace a toolbox if you accidentally delete one from your workspace.

Using Toolboxes

You can open, close, move, reshape, minimize, and maximize toolboxes on your screen. You can also delete toolboxes from your workspace, import toolboxes from other workspaces, and create new toolboxes.

Cascading toolboxes give you access to **Terramodel** commands without cluttering your screen. When you choose a special cascade button, a toolbox with additional command choices cascades from the button. You can click any button in the cascaded toolbox to execute a command, or you can open the cascaded toolbox by tearing it away from the button.

Parts of a Toolbox

Figure 18-1 shows the parts of a toolbox.

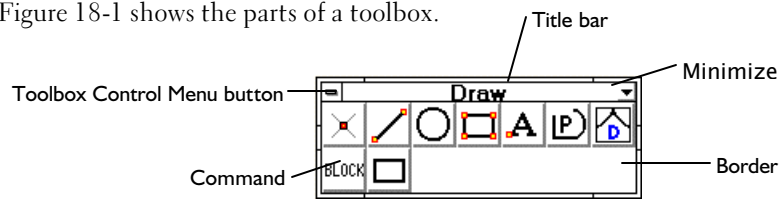


Figure 18-1 Typical Toolbox

Point to a button on the toolbox to see its name and function.

Table 18-1 Parts of a Toolbox

Toolbox part	Function
Toolbox Control Menu Button	Displays the Toolbox control menu.
Title Bar	Displays the toolbox name; moves the toolbox.
Minimize Button	Shrinks the toolbox to an icon.
Border	Reshapes the toolbox.
Command Button	Executes a command.

Using the Toolbox Control Menu

Use the Toolbox control menu on any of the toolboxes to access ToolBox controls: command buttons, toolboxes, workspaces, on-line help, about ToolBox, and exit.

Manipulating Toolboxes on the Screen

You can move, close, reshape, minimize and maximize toolboxes on your screen with your mouse.



Sometimes toolboxes become hidden under other toolboxes. If you cannot see a toolbox, move or close other toolboxes until you find the one you want.

To move a toolbox:

- ▶ Drag the title bar.

To close a toolbox:

- ▶ Double-click the Toolbox Menu Button.



You can open closed toolboxes using the Toolboxes dialog box.

To reshape a toolbox:

- ▶ Drag the border to stretch a toolbox to a different shape.

To minimize a toolbox to an icon:

- ▶ Click the minimize button. Minimized toolbox icons remain in front of the Terramodel window.

To restore a minimized toolbox to the screen:

- ▶ Double-click the icon of the toolbox you wish to restore.

The Toolboxes Dialog Box

You can use the *Toolboxes* dialog box to open and close toolboxes, create new toolboxes, import toolboxes from another workspace, rename toolboxes, and delete toolboxes from the current workspace.

To open the *Toolboxes* dialog box:

- Open the Toolbox control menu.
- Select **Toolboxes**. The *Toolboxes* dialog box appears.

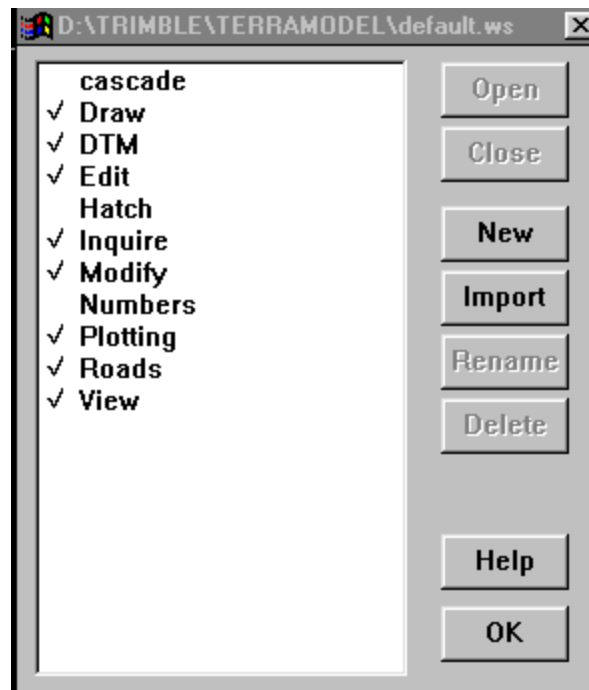


Figure 18-2 Toolboxes Dialog Box

The name of your current workspace appears in the title bar of the dialog box. All of the toolboxes in your current workspace appear in the list at left. The open toolboxes are marked with a check.

The following table outlines the controls appearing on the *Toolboxes* dialog box. Use this table as a guide to more detailed information about these functions:

Table 18–2 Toolboxes Dialog Box Controls

Control	Function
Open	Opens a selected toolbox. See below.
Close	Closes a selected toolbox. See page 604.
New	Creates a new toolbox in the workspace. See page 604.
Import	Imports a toolbox from another workspace. See page 605.
Rename	Renames a selected toolbox. See page 605.
Delete	Permanently deletes a selected toolbox from your current workspace (and disk). See page 606.
Help	Opens ToolBox Help.
OK	Closes the dialog box.

Open

A workspace may contain closed toolboxes. Opening a toolbox adds it to the screen.

To open a toolbox:

- Open the *Toolboxes* dialog box.
- Select the name of the toolbox you want to open.
- Choose the Open button, or double-click the toolbox name. The toolbox will open.
- Choose the OK button to exit the dialog box.



Sometimes toolboxes become hidden under other toolboxes or behind the Toolboxes dialog box. If you cannot see a toolbox, move or close other toolboxes or the dialog box until you find the one you want.

Close

You can close toolboxes from the *Toolboxes* dialog box. Closing a toolbox removes it from the screen, but does not permanently remove it from your workspace.

To close a toolbox:

Either:

- ▶ Double-click the Toolbox Menu Button of the toolbox you want to close,
- or,
- ▶ In the *Toolboxes* dialog box, select the name of the toolbox you want to close.
- ▶ Choose the Close button, or double-click the toolbox name. The toolbox will close.
- ▶ Choose the OK button to exit the dialog box.

New

You can create new toolboxes for your own design and drafting needs.

To create a new toolbox:

- ▶ Open the *Toolboxes* dialog box.
- ▶ Choose the New button.

A new toolbox appears in your workspace. ToolBox assigns a unique name to the toolbox such as “noname01”. You will want to rename the toolbox to a name that means something to you. To do so:

- ▶ Select the default name for your new toolbox.
- ▶ Choose the Rename button.
- ▶ Enter a name for the toolbox.
- ▶ Choose the OK button.

Import

You can import toolboxes created in another workspace into the current workspace.

To import a toolbox from another workspace:

- Open the *Toolboxes* dialog box.
- Choose Import. The *Import Toolboxes from Workspace* dialog box will appear.
- Select the workspace containing the toolbox you want to import.
- Choose the OK button. The *Import Toolboxes* dialog box will appear.
- Choose the names of the toolboxes you want to import. (Hold down the SHIFT key to select a group or the CTRL key to select individual names.)
- Choose Import. The toolboxes you chose will be imported into your current workspace.



Toolbox names cannot duplicate toolbox names already in your current workspace. If you want to import a toolbox with a duplicate name, you must rename the toolbox in the current workspace or in the workspace from which you are importing.

Rename

You can rename a toolbox at any time.

To rename a toolbox:

- Open the *Toolboxes* dialog box.
- Select the toolbox you want to rename.
- Choose the Rename button.
- Enter a new name for the toolbox.
- Choose the OK button.

Delete



Use caution when deleting Toolboxes! When you delete a Toolbox, it is deleted permanently from your computer's hard disk.

To permanently delete a toolbox from the workspace:

- ▶ Open the *Toolboxes* dialog box.
- ▶ Select the name of the toolbox you want to delete.
- ▶ Choose the Delete button. The toolbox is deleted from your workspace.
- ▶ Choose the OK button to exit the dialog box.

Using Command Buttons

When you start ToolBox, toolboxes appear on the screen. Each toolbox is a collection of *command buttons*. Command buttons provide instant access to **Terramodel** commands.

Click a command button to execute that command. A command button can have two **Terramodel** commands defined—one for the left mouse button and one for the right mouse button.

You can copy or move a command button within a toolbox or to a different toolbox. You can delete command buttons or create new command buttons.

Using the Command Button Editor you can edit the title, bitmap icon, notes, left mouse button command, and right mouse button command. These options allow you to completely customize your command buttons.

Using the Bitmap Editor, you can create new bitmap icons for your custom command buttons.

Each command button includes:

- A command button title.
- A bitmap icon.
- Pop-up notes text.
- Left mouse button command.
- Right mouse button command.

You can modify each of these properties.

Executing a Command

Many command buttons have one command defined for the left mouse button and another command defined for the right mouse button.

To execute a command, click a command button with the appropriate mouse button.

Viewing the Command Button Title

If you move your mouse cursor over a command button and pause, the command button title displays in a yellow box. A vertical line divides the left mouse button command from the right mouse button command. For many buttons, this will be all the help you need to recognize the command.

Changing the Contents of Your Toolboxes

You can move, copy, and delete command buttons from toolboxes to create custom toolboxes.

Moving a Command Button

You can move command buttons within the same toolbox or to another toolbox.

To move a command button:

- Press **SHIFT+Right mouse button** over the command button you want to move. (Notice that the cursor tells you what the action is.)
- Drag the command button to another toolbox, or drag the command button to a different location in the same toolbox.
- Release the mouse button where you want to drop the command button.
- Save your workspace.

Copying a Command Button

You can copy command buttons to another toolbox or within the same toolbox.

To copy a command button:

- ▶ Press CTRL+Right mouse button over the command button you want to copy. (Notice the cursor tells you what the action is.)
- ▶ Drag the command button to another toolbox, or drag the command button to different location in the same toolbox.
- ▶ Release the mouse button where you want to drop the command button.
- ▶ Save your workspace.

Deleting a Command Button

You can delete a command button from a toolbox by moving it out of the toolbox and dropping it outside the toolbox.

To delete a command button:

- ▶ Press SHIFT+Right mouse button over the command button you want to remove. (Notice the cursor tells you what the action is.)
- ▶ Drag the command button away from the toolbox and release the mouse button.

ToolBox asks you: “OK to delete button?”

- ▶ Click OK to delete the button.
- ▶ Save your workspace.

Editing a Command Button

You can change all five command button properties: title, bitmap icon, pop-up notes text, left mouse command, and right mouse command.

To edit a command button:

- Press SHIFT+CTRL+Left mouse button over the command button you want to edit. (Notice the cursor tells you what the action is.)
- The *Edit Command Button* dialog box appears, where you can make your changes.

The Edit Command Button Dialog Box

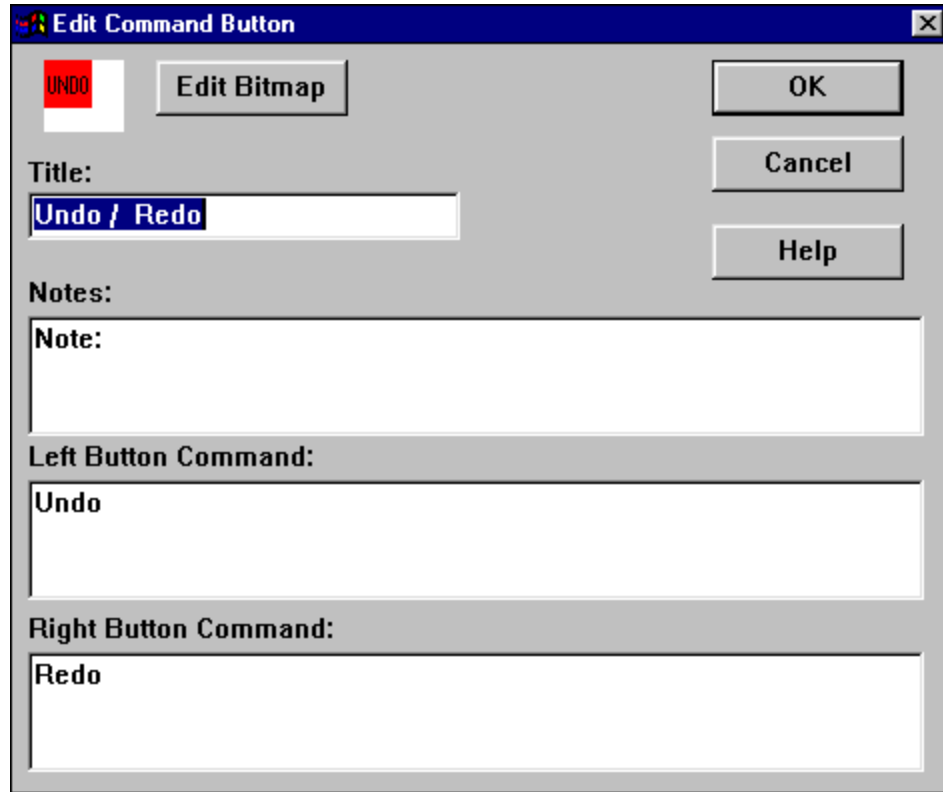


Figure 18-3 Edit Command Button Dialog Box

Changing the Bitmap Icon

To change the bitmap icon:

- Choose the Edit Bitmap button.

The Bitmap Editor appears. Use the bitmap editor to edit the bitmap icon. See page 618.

Changing the Command Button Title

The title is the yellow label that appears when you point to a command button and pause. Command buttons that have both right and left mouse buttons defined list the two actions separated by a vertical bar, for example: Undo | Redo. You may want to use this convention for your custom button titles.

To change the command button title:

- Enter new text in the Title box. The length of the text is limited to 256 characters.
- Choose the OK button to accept your changes.

Changing the Notes

To change the notes:

- Enter new text in the Notes box.
- Choose the OK button to accept your changes.

Changing the Left or Right Mouse Button Command

To change the left or right mouse button command:

- Enter a new command in the Left Button Command box or the Right Button Command box or both.



You must use ToolBox command syntax. See “ToolBox Command Syntax”, page 615.

- Choose the OK button to accept your changes.

Creating a New Command Button

You can create your own command buttons using the *Edit Command Button* dialog box. You can control all five command button properties: title, bitmap icon, pop-up notes text, left mouse command, and right mouse command.

To create a new command button:

- From the Toolbox Control Menu of the toolbox in which you want the new button to appear, choose New Button. The *Edit Command Button* dialog box will appear.
- Enter text in the Title box. The length of the text is limited to 256 characters.
The title is the yellow label that appears when you point to a command button and pause. Command buttons that have both right and left mouse buttons defined list the two actions separated by a vertical bar, for example Undo|Redo. You may want to use this convention for your custom button titles.
- Enter text in the Notes box if you wish.
- Enter commands in the Left Button Command box and the Right Button Command box. You must use ToolBox command syntax.
- Choose the Edit Bitmap button to create a bitmap icon. The ToolBox Bitmap Editor will appear.
- Use the Bitmap Editor to design the bitmap icon. See “Using the ToolBox Bitmap Editor”, page 616, for more information.

When you have finished creating your icon, choose Exit from the FILE menu of the Bitmap Editor to return to the *Edit Command Button* dialog box.

- Choose the OK button.

Viewing Notes

You can create your own notes for command buttons. The notes can act as quick help.

To view command button notes:

- Position the mouse pointer over the command button. The cursor tells you what the command button’s action is.

- Press and hold the CTRL key.
- Click and hold the left mouse button over the command button. The notes text displays as long as you hold down the mouse button.

Using the Command Toolbox

The Command toolbox contains command buttons for additional **Terramodel** commands. To use any of the command button which it contains, import this toolbox and copy or move command buttons from it into your own toolboxes.

Using the Icon Toolbox

The Icon toolbox has additional command buttons that have no commands defined. Use or edit these bitmap icons to create your own buttons for custom commands.

Using Cascade Buttons

Some command buttons cascade toolboxes. This lets you use tools “behind” the command button on top. A small white square in the bottom right corner of the button identifies a cascading command button.

Cascade buttons allow you to keep one main toolbox that ties to a set of cascading toolboxes. The main toolbox can contain a large number of commands behind a small number of buttons. You can keep often-used commands on top, and access other command buttons by cascading or tearing away the toolboxes behind the common commands.

Creating a Cascade Button

You can link any toolbox to a command button in another toolbox. The linked toolbox will cascade from the button when you click both mouse buttons.

In the default workspace, several toolboxes are linked to buttons in the ToolBox toolbox. Look for a white square in the lower right corner of a command button to identify a button with a cascading toolbox.

To create a cascade button:

- ▶ In the toolbox you want to link, press SHIFT+CTRL+Right mouse button over any command button. Notice the cursor tells you what the action is.
- ▶ Drag the command button to another toolbox.
- ▶ Release the mouse button. The button is copied to the other toolbox and becomes a cascade button.

Using a Cascade Button

To cascade a toolbox:

- ▶ Click both the left and right mouse buttons over the button you want to cascade. Click the left or right mouse button over the command button you want to use.

To cancel the cascade:

- ▶ Click the left mouse button anywhere outside the toolbox.

Tearing Off a Toolbox

You can tear off the toolbox cascaded from a button and place it on your screen as a separate toolbox.

To tear off a cascaded toolbox:

- ▶ Click both the left and right mouse buttons over the command button. The linked toolbox will cascade from the command button.
- ▶ Click the left mouse button on the toolbox title bar and drag the toolbox to a new position on the screen. The toolbox will stand alone in the new location.

Unlinking a Cascade Button

You can remove the link between a cascade button and a toolbox. This does not remove the button from either toolbox, but simply cancels the link.

To unlink a toolbox from a cascade button:

- Press SHIFT+CTRL+Left mouse over the cascade button you want to change. The *Edit Command Button* dialog box appears. *Float to Top* and *Unlink* are the only available options.
- Choose the Unlink button.
- Choose the OK button. The link between the button and the toolbox is removed.

ToolBox Command Syntax

ToolBox command text is limited to 256 characters.

ToolBox accepts any command (or TML) recognized by Terramodel. Enter exactly what you would type at the command prompt in the Left Button Control or Right Button Control edit text box in the *Edit Command Button* dialog box. See “Using the Terramodel Macro Language”, later in this chapter, for more information.

Floating Command Buttons

The Float to Top option causes the last picked button in a cascaded toolbox to become the button displayed in the parent toolbox. If Float to Top is off, the last used button does not move to the current toolbox. Use the Float to Top option to make your favorite command in the toolbox the cascade button.

Floating Another Button to the Top Position

To float another button to the top position:

- Press SHIFT+CTRL+Left mouse over the cascade button you want to change. The *Edit Command Button* dialog box appears. *Float to Top* and *Unlink* are the only available options.

- Check the Float to Top box.
- Choose the OK button.
- Click both mouse buttons over the cascade button.
- In the cascaded toolbox, choose the command button you want to float to the top. The button becomes the cascade button.

Keeping the New Button on Top

To keep the new button permanently in the top position:

- Press SHIFT+CTRL+Left mouse over the cascade button.
- The *Edit Command Button* dialog box appears.
- Remove the check from the Float to Top box.
- Choose the OK button.



If you don't turn off Float to Top, each time you pick another button in the cascaded toolbox, that button will float to the top position.

Using the ToolBox Bitmap Editor

ToolBox uses 24x24 pixel 16-color or monochrome bitmaps for command button images. You can create these bitmap icons with the ToolBox Bitmap Editor.

The Bitmap Editor allows you to edit bitmap icons, or you can use the Grab feature to capture a region of the screen into the editor.

Opening the Bitmap Editor

To open the Bitmap Editor:

- From the Toolbox Control Menu, choose New Button,
or
Press SHIFT+CTRL+Left mouse button over the command button you want to edit.
- From the *Edit Command Button* dialog box, choose Edit Bitmap. The Bitmap Editor appears.

The Bitmap Editor

Use the Bitmap Editor (*Edit Bitmap* dialog box) to create new icons for new or existing command buttons. You may create new icons from scratch, or edit existing ones.

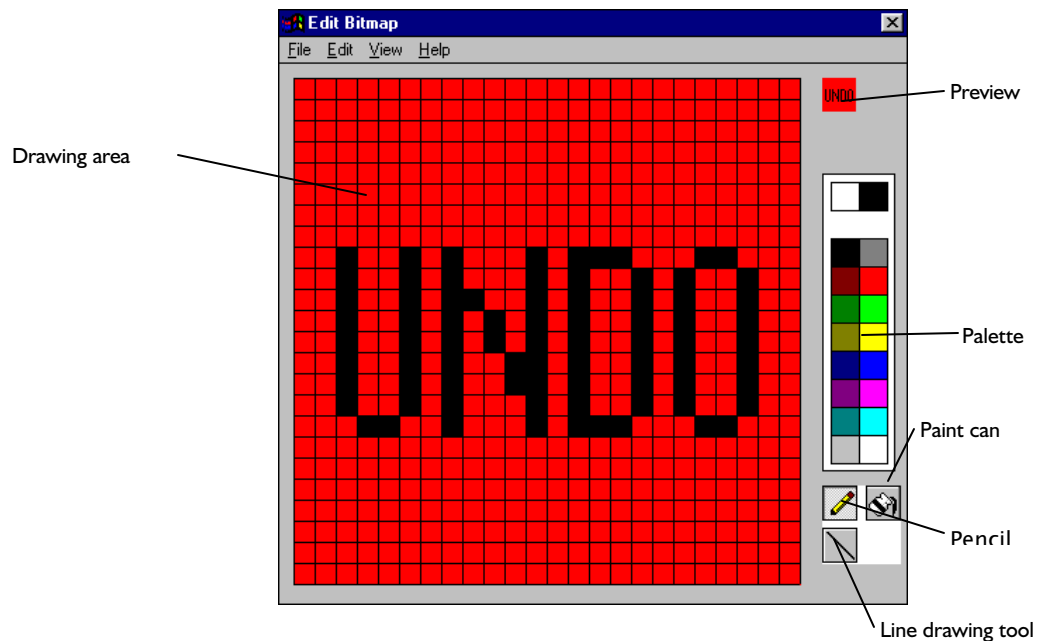


Figure 18-4 Toolbox Edit Bitmap Dialog Box

Table 18-3 Bitmap Editor Description

Menus, Screen & Tools	Description
File Menu	Includes Import Bitmap, Export Bitmap, Browse Workspace, and Exit Bitmap Editor commands.
Edit Menu	Includes Grab, Copy, Clear, Paste, and Undo commands.
View Menu	Includes Zoom In and Zoom Out commands.
Help Menu	Opens Toolbox Help.

Menus, Screen & Tools	Description
Drawing Area	An enlarged view of your bitmap icon. Edit your bitmap icon in this area.
Palette	Contains the colors you can use in your bitmap icon. The two squares above the palette show the colors loaded in your left and right mouse buttons.
Button Preview	Displays your bitmap icon in actual size.
Pencil Tool	Use to edit the bitmap icon pixel-by-pixel.
Paint Can Tool	Fills a closed shape or area with the selected color.
Line Drawing Tool	Draws a straight line.

Using the Editing Tools

The Bitmap Editor has three editing tools:

- The pencil tool edits the bitmap icon pixel-by-pixel.
- The paint can tool fills a closed shape or area with the selected color.
- The Line tool draws straight lines.

To edit the bitmap icon:

- Click the pencil, paint can, or line tool.
- Click the Palette to load the mouse buttons with color. You can load both the left and right mouse buttons with color. The two boxes immediately above the Palette show which colors are currently loaded.
- Click or drag either mouse button in the drawing area to draw the bitmap icon.

Using the Grab Feature

The Bitmap Editor's Grab feature allows you to capture a region on your screen for use in a button.

To grab a screen region:

Make sure the region you want to capture is visible on your screen.

Either:

- ▶ Choose New Button from the Toolbar Control Menu of the toolbox in which you want the new command button to appear.

or,

- ▶ Press SHIFT+CTRL+Left mouse button over the command button whose icon you want to replace. The *Edit Command Button* dialog box appears.
- ▶ Choose Edit Bitmap. The Bitmap Editor appears. The current icon for the selected command button appears in the drawing area.
- ▶ From the **EDIT** menu, choose **Grab**. A square will replace your mouse pointer on the screen. The square represents a 24x24 pixel area on your screen.
- ▶ Aim the square over the screen area you want to capture, then click. The image is captured in the drawing area.

Saving the Bitmap to a File

You can save the bitmap icon you created within ToolBox as a .BMP file if you want to use it for some other purpose.

To save a bitmap file:

- ▶ From the **FILE** Menu of the Bitmap Editor, choose **Export Bitmap File**. The *Export Bitmap File* dialog box appears.
- ▶ Enter a name for the bitmap file.
- ▶ Choose the OK button. ToolBox saves the bitmap in 24x24, 16-color format.

Exiting the Bitmap Editor

When you finish editing your bitmap icon, exit the Bitmap Editor.

To exit the Bitmap Editor:

- From the **FILE** menu, choose **Exit**. The bitmap icon is applied to your current button and saved as part of your current workspace.

Creating a New Bitmap Icon from Scratch

When you wish to design an icon yourself, rather than basing it on an existing icon, follow the steps below.



ToolBox supplies the bevel edges that make your bitmap icon look like a three-dimensional push-button—you do not have to draw them.

To create a new bitmap icon:

- From the Toolbox Control Menu of the toolbox in which you wish the new icon to appear, choose **New Button**.
- From the *Edit Command Button* dialog box, choose **Edit Bitmap**. The Bitmap Editor appears.

You may design on the gray background, or clear the gray background and start. To clear the drawing area, choose **Clear** from the **EDIT** menu.
- Create your bitmap drawing.



You may save your bitmap to a file if you wish. See “Saving the Bitmap to a File”, page 620.

- From the **FILE** menu of the Bitmap Editor, choose **Exit**. In the *Edit Command Button* dialog box, make any changes needed to your title, notes, and commands for this command button. See The *Edit Command Button* dialog box, page 610, for more details.
- When you are finished, choose the **OK** button.

Creating a New Bitmap Icon Using an Existing Bitmap

There are two methods for using an existing bitmap as the basis for a new one:

1. You can browse through the available workspaces for a bitmap icon already in use by a button, and edit it.
2. You can import a bitmap created outside of ToolBox.

Importing a Bitmap Icon from Another Workspace

To import a bitmap icon from another workspace:

- From the Bitmap Editor **FILE** menu, choose **Browse Workspace**. The *Import Bitmap from Workspace* dialog box appears.
- Select the workspace you want to use. The *Select Bitmap* dialog box appears, displaying all of the bitmap icons in the selected workspace.
- Use the arrow buttons at right to scroll through the bitmap icon collection.
- Click the bitmap icon you want to use. A copy of the bitmap icon appears in the Bitmap Editor.
- Make your changes to the bitmap icon. The original bitmap icon will remain unchanged. ToolBox creates a copy of the image and saves it with your current workspace.



You may save your bitmap to a file if you wish. See “Saving the Bitmap to a File”, page 620.

- From the **FILE** menu of the Bitmap Editor, choose **Exit**. In the *Edit Command Button* dialog box, enter your title, notes, and commands for this command button. See The *Edit Command Button* Dialog Box, page 610, for more details.
- When you are finished, click OK.

Importing a Bitmap Created Outside of ToolBox

To import a bitmap created outside of ToolBox:

- From the Bitmap Editor **FILE** menu, choose **Import Bitmap**. The *Import Bitmap File* dialog box appears.

-
- Select a bitmap file to edit.
 - Choose the OK button. A copy of the bitmap will appear in the Bitmap Editor.
 - Make changes to the bitmap. The original bitmap icon will remain unchanged. ToolBox creates a copy of the image and saves it with your current workspace.



You may save your bitmap to a file if you wish. See “Saving the Bitmap to a File”, page 620.

- From the **FILE** menu of the Bitmap Editor, choose **Exit**. In the *Edit Command Button* dialog box, enter your title, notes, and commands for this command button. See The *Edit Command Button* Dialog Box, page 610, for more details.
- When you are finished, choose the OK button.

Customizing the Terramodel Toolbar

You can assign your most commonly used commands to the buttons at the right end of the main **Terramodel** Toolbar. You can even create and assign pictures (bitmaps) to the buttons to remind you of the commands that are mapped to them.

The TMODWIN.INI file contains a section called [Toolbar] which controls the visibility of the Toolbar and maps commands to the Toolbar buttons.

You can assign a bitmap to a button by adding a comma and the name of the bitmap file after each command name. **Terramodel** will convert the bitmap to the appropriate size to fit on the button. Avoid large bitmaps, since they typically will not be legible when reduced to button size. The best way to create a bitmap is using the ToolBox Bitmap Editor, described beginning on page 616.

The button-mapping takes effect the next time you re-start **Terramodel**.

The Toolbar

See Chapter 2 of this user guide, or the Terramodel online help, for detailed information about the Terramodel toolbar and toolbar toggle commands.

Customizing Special Function Keys

You can change how commands are assigned to function keys [f1]-[f12] by editing the [FunctionKeys] section of the TMODWIN.INI file. Table 18-4 lists how this section is mapped when you first install **Terramodel**.

Table 18-4 Function Keys

Key	Command	Key	Command
F1	HELP (Controlled by Windows.)	F7	gridset -1

F2	point	F8	cursor -1
F3	set	F9	snapset -1
F4	pline	;F10	MENU BAR (Controlled by Windows.)
F5	id	F11	cursor -2
F6	search	F12	undo

By pressing the SHIFT, CTRL, or SHIFT+CTRL keys simultaneously with a function key, you can access additional commands. Table 18–5 lists the commands that are mapped to the function keys when you press one of these other keys in conjunction with a function key.

Table 18–5 SHIFT, CTRL and ALT Function Keys

# SHIFTed function keys	# CTRL function keys	#CTRL+SHIFT function keys
SF1=getpoint.int	CF1=getpoint.bb	CSF1=
SF2=getpoint.mid	CF2=getpoint.bd	CSF2=
SF3=getpoint.mid2pt	CF3=getpoint.cen	CSF3=
;SF4=TILE (This is controlled by Windows.)	;CF4=CLOSE ACTIVE VIEW WINDOW (This is controlled by Windows.)	CSF4=
:SF5=CASCADE (This is controlled by Windows.)	CF5=getpoint.dd	CSF5=
SF6=getpoint.ne	;CF6=NEXT VIEW WINDOW (This is controlled by Windows.)	CSF6=
SF7=getpoint.offset	CF7=getpoint.end	CSF7=

# SHIFTed function keys	# CTRL function keys	#CTRL+SHIFT function keys
SF8=getpoint.oi	CF8=getpoint.ins	CSF8=
SF9=getpoint.perp	CF9=	CSF9=
;SF10=MENU BAR (This is controlled by Windows.)	;CF10=MENU BAR (This is controlled by Windows.)	;CSF10=MENU BAT (This is controlled by Windows.)
SF11=getpoint.pi	CF11=	CSF11=
SF12=getpoint.xy	CF12=	CSF12=

Using the Terramodel Macro Language

The **Terramodel** Macro Language (TML) lets you customize and enhance the capabilities of **Terramodel**. Anyone with programming experience and a basic knowledge of the C programming language can write and edit TMLs.

Most of the commands for the COGO, Sewer and Hydro modules are actually TMLs that you can alter to obtain exactly the type of information you're looking for. By editing these TMLs, you can draw the graphics for these modules in the format your company uses.

Where to Learn About Programming with TMLs

When you installed **Terramodel**, Windows created a new folder named **Terramodel**. Included in that folder is an icon named TML Help.

By clicking the icon you can view information about programming in TML. You can also print any topic listed in the help screen by choosing **Print Topic** from the TML Help **FILE** menu.

The best source of information is to review the TML commands that are included with the program. Many times you can find existing example of something very close to what you might need.

TML Basics

What is a TML Macro?

A TML macro is a named group of statements and function calls enclosed by braces. A macro is stored in a sequential ASCII data file on your hard disk. The macro file contains statements that perform tasks that are specific to your company or are not yet implemented by **Terramodel**.

A TML macro can do almost anything that can be done by a **Terramodel** programmer. A TML macro can prompt you for character or graphic input, scan databases and generate reports, create and modify graphic elements, and read and create database elements from external data files.

Macro statements must be written into an ASCII file using a system editor such as *Notepad*. Each macro file must have a .TML file extension, and must reside in the current default directory or in one of the directories in the TMWPATH environment variable.

TMLs and Windows

TML macros use dialog boxes and command bars as the main type of interface with Windows. Dialog boxes and command bars can contain special controls (like list boxes, check boxes, and radio buttons) that each have their own functions for operation and data manipulation. Trimble provides several different examples of pre-made dialog boxes that TML writers can use. If you desire to do more than make small adjustments to existing TML commands you will need to get a resource editor designed to work with Windows. The resource editor will enable you to design the user interface portion of your TML graphically.

TML Function Reference

TML includes a large number of built-in functions that allow you to interface with the operating system and **Terramodel**. These internal routines can be used to query, add to or modify an object, and do everything from taking the square root of a number to creating **Terramodel** objects. For information on function references, see TML Function Reference in the TML Help system.

Terramodel Utility Functions

The TML utility functions can be used to retrieve system settings and perform geometric calculations. The functions that are available to you include:

Managing layers	User-defined linetypes
Group ID tags	Searching
Displaying graphics	Geometry functions
Miscellaneous string functions	Alignment functions
Managing attribute data	Special application functions
Coordinate transformation routines	De-bugging
Error messages	Surface model access

For more information about utility functions, see *Terramodel Utility functions* in the TML Help system.

Index

(Tc), 395

.LIN file format, 150

—A—

Active alignments, 125
Albers Equal-Area Conic Projection, 517
American Polyconic Projection, 517
AND NOT select control, 244
AND select control, 244
Angle control, 218
Angle snap modes, 258, 259, 260, 261
Aperture, 52
Attribute text, 110
Attributes, 124
Azimuthal Map projection, 479
Azimuthal Projection, 517

—B—

Base Sheet, creating, 428
Basins, multiple, 417
Bearing control, 219
Bearings
 Snap modes, 254
Bitmap Editor. *See* IconTOOL Bitmap Editor
Bitmap icon
 Replacing, 611
Bitmaps. *See also* Icons
Blank Sheets
 Creating, 442
Blocks, 106–15

Commands, 114
Content of, 113
Creating, 114
Exploding, 114
Included with TERRAMODEL, 181
Title, 181
Uses for, 113
Branches, 394

—C—

CAD Controls. *See* Controls, Custom
Callouts, 159
Cartographer, 518
Cascade buttons. *See* Command buttons
Central Meridian, 518
Client Data Dialog Box, 453
Color control, 210
Color select control, 234
Color Selection Dialog Box, 145
Color-mapping
 Creating, 146
Colors
 Mapping colors to pens, 146
 Working with, 139–46
Command bar, 76
Command buttons, 607–13
 Cascade buttons, 613
 Creating, 613
 Unlinking, 615
 Using, 614
 Changing commands, 611
 Changing notes, 611
 Changing the title, 611
 Copying, 609
 Creating, 612
 Deleting, 609
 Editing, 610

- Executing a command, 608
 - Floating, 615
 - Keeping new on top, 616
 - Icons
 - Replacing, 611
 - Moving, 608
 - Notes
 - Viewing, 612
 - Viewing the title, 608
 - Command syntax
 - IconTOOL, 615
 - Command toolbox, 613
 - Configuring TERRAMODEL, 44–53
 - Conformal Projection, 518
 - Conic Map Projection, 483
 - Conic Projection, 518
 - Control menus, 58
 - Options, 58
 - Controls
 - Button, 63
 - Dialog box, 59
 - Windows, 60
 - Controls, Custom
 - CAD, 209–13
 - Color, 210
 - Edit, 210
 - Hatch, 212
 - Layer, 211
 - Linetype, 211
 - Text size, 213
 - Locate, 214, 215
 - Locate object, 214, 215
 - Number, 216
 - Integer number, 216
 - Point number, 216
 - Real number, 216
 - Select, 230
 - AND, 244
 - AND NOT, 244
 - Color, 234
 - EQUALS, 243
 - Expression, 240–43
 - Layer, 235
 - Linetype, 236
 - Name, 237
 - NearLine, 237
 - NOT, 244
 - Or, 243
 - Points, 238
 - Previous, 243
 - Type, 239
 - View, 239
 - Window, 240
 - XOR, 244
 - Value definition, 217–27
 - Angle, 218
 - Bearing, 219
 - Coordinate, 221
 - Distance, 223
 - Elevation, 224
 - Hide and pick, 224
 - Offset, 224
 - Station, 225
 - Vertical angle, 226
 - Converting Coordinates, 458, 506
 - Coordinate control, 221
 - Coordinate Conversion Routines, 457
 - Coordinate scroll, 74
 - Coordinate System, 518
 - Copying Data Between Projects, 349
 - , 549
 - Corrections, 316
 - Cross-sections, 359
 - Cylindrical (Mercator) Map Projection, 481
 - Cylindrical Projection, 519
- D—**
- Data Tags, 339
 - Datum, 519
 - Datums, 468
 - DeKalb Rational Method, 409
 - Design Settings, Roadway, 357
 - Detail Pane, 310
 - Dialog boxes, in general, 58
 - Digitizer, configuring, 582
 - Digitizer, creating a new menu, 586
 - Digitizer, map orientation, 591
 - Digitizer, orienting, 584

Display Pane, 302
 Distance
 Snap modes, 256
 Distance control, 223
 Dragging on/off, 52
 Drawing Sheet Components
 Adjusting, 444
 Drawing Sheets
 Creating, 438–41
 Recreating, 441
 DTM
 Adding breaklines to, 269
 and Links, 264
 Definition, 263
 DtmEdge
 Changing links with, 268
 DTMEdge command, 273
 Dynaviews, 119–23
 Advantages of, 122
 and Object referencing, 121
 Changing, 122
 Commands, 123
 Creating, 123
 Displaying objects through, 120
 Selecting, 123
 Storing, 119
 Working with, 179–80

—E—

EAT codes, 160
 Eccentricity, 519
 Edit Command Button Dialog Box, 610
 Elevation control, 224
 Ellipsoid, 470, 519
 Embedded Attribute Text, 160
 Equal Area Projection, 519
 EQUALS select control, 243
 Equatorial Radius, 520
 Errors, RDE, 327
 Escape count, 51
 Expression select control, 240–43

—F—

False Easting, 520
 False Northing, 520
 Field Data module, 27
 File locking, 52
 Flattening Factor, 520
 Floating command buttons, 615
 Flow rate calculation methods, 395
 Q Summation, 395
 Rational method, 395
 Fonts
 Included with TERRAMODEL, 197
 Fonts, text, 156

—G—

General Information Dialog Box, 451
 Geodesy, 520
 Geodetic, 520
 Geodetic Reference System, 521
 Geoid, 521
 GIS, 457
 GPS, 457
 Graphic scales, 183

—H—

Hardware lock, installing, 42
 Hatch (pattern) control, 212
 Hatch Pattern Dialog Box, 176
 Hatch patterns
 Creating custom, 177
 Example, 178
 Working with, 175–76
 Hide and pick control, 224
 Horizontal Alignment Offsets, 360
 Horizontal alignments, 115–18, 356
 Commands, 117
 Creating, 117
 Using with vertical alignments, 116
 Horizontal angle modes, 220
 Hotine Oblique Mercator Projection, 502,
 521
 Hydrology
 Concepts, 406

Terminology, 406
 Hydrology commands
 Using, 405–18
 Hydrology module, 405–18

— I —

Icon toolbox, 613
 Icons
 Creating, 621–23
 Importing, 622–23
 IconTOOL, 595–623
 IconTOOL Bitmap Editor, 616–23
 Description of, 618
 Editing tools, 619
 Exiting, 621
 Grab feature, 620
 Opening, 617
 Saving bitmaps to file, 620
 IconTOOL Command Syntax, 615
 Import Cut/Fill Volumes, 392
 Import/Export Commands, 280
 Initialization file, 53
 installation, 33
 Integer number control, 216
 International Survey Foot, 521
 internet address, 24

— K —

Key combinations, 61
 Dialog box, 62
 Editing, 61
 System, 61

— L —

Labels
 Controlling placement on drawing
 sheets, 441
 Lambert Conformal Conic Projection, 499,
 521
 Latitude, 522
 Latitude and Longitude, 463
 Layer color control, 72
 Layer control, 211

Layer Offset Dialog Box, 136
 Layer select control, 235
 Layer selection control, 72
 Layer Selection Dialog Box, 72, 138
 Layer Setup Dialog Box, 71, 132
 Layers
 Assigning colors, 131
 Moving objects to other layers, 138
 Working with, 130–38, **Error! Not a
 valid bookmark in entry on page 139**
 Least Squares, 328
 Library workspace, 599
 Linetype
 Changing by layer, 147
 Changing by object, 148
 Linetype control, 211
 Linetype Dialog Box, 73, 148
 Linetype file
 Example, 152
 Loading, 147
 Linetype select control, 236
 Linetype selection control, 73
 Linetypes
 .LIN file format, 150
 Custom
 Creating, 149
 Types of, 149
 Using symbols in, 151
 Included with TERRAMODEL, 181,
 185
 Working with, 147–52
 Link Buffer, 268
 Link Settings Dialog Box, 264
 Link Step, 268
 Locate controls. *See* Controls, Custom. *See*
 Controls, Custom
 Locate object control, 214, 215
 Locked Project, 343
 Longitude, 522

— M —

Macro
 Defined, 628
 Macro Directory, 50

Map Projections, 459
 Marker size, 52
 Mass Haul Analysis, 370
 Master Project, 341
 Master Sheet, creating, 430
 Material, defined, 355
 Max Alpha Pts, 51
 Max objects, 50
 Maximum Edge Angle, 266
 Maximum Edge Distance, 265
 Menu bar, 64
 Menu conventions, 65
 Mercator Projection, 522
 Meridian, 522
 Meridians, 466
 Message Scroll, 77
MhImport, 392
 Mirrored Text, 159
 Modules, TERRAMODEL, 25
 Mouse conventions, 22
 Mouse pointer shapes, 63
 Multilayer DTMs, 274
 Multiple Target Method, 314

—N—

NAD-27, 476
 NAD-83, 477
 NADCON Software, 523
 Name select control, 237
 NearLine select control, 237
 New Layer Dialog Box, 136
 New Project Name Dialog Box, 80
 Nodes, 393
 North American Datum of 1927 (NAD-27), 523
 North American Datum of 1983 (NAD-83), 523
 North arrows, 182
 NOT select control, 244
 Number controls. *See* Controls, Custom

—O—

Objects

Moving to other layers, 138
 Referencing to other objects, 124
 Searching for, 245
 Snap modes, 247
 Types of, 91
 Offset control, 224
 On-Line Help, 31
 Open a Project Dialog Box, 87
 Or select control, 243
 Origin Latitude, 523
 Origin Longitude, 524

—P—

P3Pad, 527
 Palettes
 Creating, 146
 Parallel, 524
 Phases, defined, 361
 Pipe
 Concepts, 393
 Terminology, 393
 Pipe module, 370–405
 Pipe sizes
 Assigning, 398
 Pipe system
 Analysis, 404
 Design, 401
 Pipe systems, 394
 Pipes, 394
 Plan Set
 Automated Sheet Assembly, 427
 Base Sheet, creating, 428
 Blank Sheets
 Creating, 442
 Blank Sheets, Using, 427
 Commands, using, 419
 Concepts and Terminology, 422
 Configuration Settings, 424
 Graphical Editing of Label Settings, 424
 Drawing Sheet Components
 Adjusting, 444
 Drawing Sheets
 Creating, 438–41

- Recreating, 441
- Index of Drawings, Creating, 426
- Labels
 - Controlling placement on drawing sheets, 441
- Master Sheet, creating, 430
- Miscellaneous Sheets, Using, 427
- Plotbox Visibility
 - Controlling, 437
- Plotboxes
 - Creating, 431–38
 - Deleting, 437
 - Editing in Plan View, 434
 - Editing in Profile View, 436
- Rotation of Dynaviews
 - Controlling, 441
- Selecting Sheets, 443
- Sheet Insertion Point Marker
 - Placing, 443
- Sheets
 - Combining, 444
 - Deleting, 445
 - Reordering, 445
 - Viewing individual, 446
 - Viewing sheet information, 446
- Sheets, working with, 428–47
- Supported Drawing Configurations, 425
 - Alignment-Based Cross-Section Sheets, 426
 - Alignment-Based Plan and Profile Sheets, 426
 - Alignment-Based Plan Sheets, 425
 - Miscellaneous Sheets, 426
- Titles
 - Controlling placement on drawing sheets, 441
- Working with Plan Sets, 422
- Plan Set Assembly and Management Module, 419–47. *See also* Plan Set
- Plot Settings, 569–76
- Plot Settings Dialog Box, 569
- Plotbox Visibility
 - Controlling, 437
- Plotboxes
 - Creating, 431–38
 - Deleting, 437
 - Editing in Plan View, 434
 - Editing in Profile View, 436
- Plotters
 - Selecting, 560
 - Setting up, 559
- Plotting
 - Defining the area to plot, 577
 - Project files, 577
 - To a device, 577
 - To a file, 577
- Plotting devices. *See* Plotters, Printers
- Point color control, 72
- Point number control, 216
- Points, 93–95
 - Commands, 94
 - Creating, 93
 - Selecting, 95
 - Snap modes, 247–54
- Points select control, 238
- Polylines, 102–5
 - Commands, 104
 - Creating, 104
 - Makeup of, 102
 - Selecting, 105
 - Types, 102
 - Uses for, 103
- Precision, Accuracy & Significant Figures, 326
- Previous select control, 243
- Printers
 - Selecting, 560
 - Setting up, 559
- Project Data
 - Editing, 449
 - Linking to text objects, 455
- Project Data Content
 - Customizing, 455
- Project Data Editor, 449–56
- Project file locking, 52
- Project files
 - Building, 80–84
 - Closing, 86

- Creating new, 79
- Opening, 86
- Prototype
 - Creating, 88
- Retrieving, 86
- Saving, 85
- Project Location Dialog Box, 452
- Projection, 524
- Prompt if create pt in Set cmd, 53
- Prototyp project file, 51

—Q—

- Q Summation, 395
- Quick Contours, 267
- Quick Isopach Generation, 267

—R—

- Raw Data Editor, 293
- RDE – Calculation Methods, 331
- RDE Computation Settings, 311
- RDE Menu Bar, 300
- RDE Report Options, 317
- RDE settings, 311
- Real number control, 216
- Recreating Drawing Sheets, 441
- Reference Project, 342
- Reference station, 324
- Referencing objects to other objects, 124
- Remove Flat Triangles, 265
- Reports, Creating, 527
- requirements, 33
- Road
 - Concepts, 354–62
 - Design, 363
 - Multiple
 - Design, 368
 - Single
 - Design, 366
 - Terminolgy, 354–62
- Road Job Stationing, 365
- Road jobs, 357
- Road module, 354–69
- RoadJob, 365

- Roadway. *See* Road
 - Design, multiple roads, 368–69
 - Design, single road, 365–67
- Roadway, defined, 358
- Rotation of Dynaviews
 - Controlling, 441
- Rough Ref. Station Tolerances, 316
- Rough reference station, 324
- Running snap modes, 263
- Running Snap Modes Dialog Box, 263

—S—

- Save Current Project As Dialog Box, 85
- Scale of the Central Meridian, 524
- Scale Reduction Factor, 524
- Scales
 - Graphic, 183
- Script Manager, 285
- SCS Method, 407
- Search Settings Dialog Box, 245
- Secured commands, 28
- Select By Color Dialog Box, 234
- Select By Layer Dialog Box, 235
- Select By Linetype Dialog Box, 236
- Select By Name Dialog Box, 237
- Select By Point Number Dialog Box, 238
- Select By Type Dialog Box, 239
- Select By View Dialog Box, 239
- Select By Window Command Bar, 240
- Select controls. *See* Controls, Custom
- Select Plotter Dialog Box, 561
- Sentinel, TERRAMODEL, 42
- Sets, 96–101
 - Commands, 100
 - Content of, 96
 - Creating, 100
 - Selecting, 101
 - Types of, 96
 - Uses for, 98
- Sewer system
 - Analysis, 404
 - Design, 401
- Shape classes, 356
- Shape, defined, 356

- Sheet Insertion Point Marker
 - Placing, 443
 - Sheets
 - Combining, 444
 - Deleting, 445
 - Reordering, 445
 - Viewing individual, 446
 - Viewing sheet information, 446
 - Sideshot, 324
 - Single Target Method, 313
 - Skip, defined, 360
 - Smart Text, 160
 - Snap modes
 - Angle, 258, 259, 260, 261
 - DeflAngle, 258
 - ThreePt, 258
 - Bearing, 254
 - Brg+Ang, 255
 - ofLine, 255
 - Pt-Pt, 255
 - Distance, 256
 - Dist+Dist, 256, 261
 - Fact, 256, 261
 - ofLine, 256, 261
 - Pt-Pt, 257, 261
 - Radius, 257, 262
 - Point, 247–54
 - Bearing-Distance, 248
 - Center, 248
 - FactHal, 249
 - Free, 249
 - Insertion point, 250
 - Intersect, 250
 - IntHal, 250
 - Mid, 250
 - Name, 251
 - Nearest, 251
 - OffHal, 252
 - OffInt, 252
 - OffIntHal, 252
 - Offset, 251
 - Perp, 252
 - PerpHal, 253
 - PI, 253
 - Point, 253
 - Relative distance, 248
 - Xy, 254
 - Running, 263
 - StaElev, 254
 - Tangent, 254
 - Vertical, 257
 - ofLine, 257
 - Pt-Pt, 257
 - Snap Modes, 247
 - Snap modesQuadrant, 253
 - Spline types, 272
 - Staff Data Dialog Box, 454
 - Standard deviation, 334
 - Standard Errors, 316
 - Standard Parallel, 525
 - State Plane Coordinate Systems, 504, 525
 - Station control, 225
 - Station Equations, defined, 360
 - Status bar, 67
 - Subgrade, 359
 - Superelevation View, 129
 - Superelevation, defined, 359
 - Support, 31
 - Surface, defined, 355
 - Symbols
 - Working with, 173
 - System Configuration Dialog Box, 49
 - system requirements, 33
- T—**
- Tablets, supported, 593
 - Target Height Assumption, 313
 - Target/Pogo/Rod/Range Pole, 324
 - Templates, 358
 - Terramodel Macro Language
 - and Windows, 628
 - Basics, 628
 - Function reference, 628
 - Utility functions, 629
 - Terramodel Search Path, 36
 - Text, 106–12
 - Attribute text, 110
 - Commands, 111

- Creating, 111
 - Drafting, 167
 - From external files, 167
 - Selecting, 112
 - Text objects, 106
 - Working with, 156–72
- Text Borders, 159
- Text Flipping, 169
- Text fonts, 156
- Text Leader Lines, 159
- Text Objects
- Linking to Project Data, 455
- Text Orientation, 169
- Text size control, 213
- Text Style Settings Dialog Box, 170
- Text styles
- Working with, 156–72
- Titles
- Controlling placement on drawing sheets, 441
- Tml debug level, 51
- TMODWIN.INI, 53
- Tolerance Checks, 334
- Toolbar
- Customizing, 624
- Toolbar, **TERRAMODEL**
- Controls, 68
- Toolbar, **TERRAMODEL**, 68
- Toolbox auto start, 53
- Toolboxes, 600–606
- Changing the contents of, 608
 - Closing, 601, 604
 - Command, 613
 - Control menu, 601
 - Creating, 604
 - Deleting, 606
 - Icon, 613
 - Importing from another workspace, 605
 - Minimizing, 601
 - Moving, 601
 - Opening, 603
 - Parts of, 600
 - Renaming, 605
 - Reshaping, 601
 - Restoring, 601
 - Tearing off from another toolbox, 614
- Toolboxes Dialog Box, 602
- Training, 32
- Transverse Cylindrical Projection, 485
- Transverse Mercator Projection, 493
- Triangle Slope, 267
- TSP, 50
- Type select control, 239
- Typographical conventions, 21

—U—

- U.S. Survey Foot, 526
- Undo size, 52
- Uninstalling, 36
- Universal Rational Method, 408
- Universal Transverse Mercator, 497
- Universal Transverse Mercator (UTM), 525

—V—

- Value definition controls, 218
- Vertical alignments, 115–18, 357
 - Commands, 117
 - Creating, 117
 - Using with horizontal alignments, 116
- Vertical angle control, 226
- Vertical angle modes, 227
- Vertical snap modes, 257
- View modes, 127–29
 - 5–8, 129
 - Plan, 128
 - Profile, 128
 - Sheet, 128
 - Xsection, 129
- View select control, 239
- Views, 56
 - Controls, 56

—W—

- website, 24
- Window select control, 240
- Workspaces, 597–99
 - Creating, 598

Default, 597
Defined, 597
Library, 599
Opening, 597
Saving, 598

World Geodetic Reference System of 1984
(WGS-84), 526

—X—

XOR select control, 244

Appendix A

TV Trees, Shrubs, and Examples

Introduction

This appendix contains descriptions and color graphics of trees, shrubs and geotypical textures available in Terramodel Visualizer as well as several examples of Terramodel Visualizer projects.

Trees, Shrubs and Geotypical Textures

Terramodel Visualizer includes a wide range of tree, shrub, and geotypical texture images that can be used in your visualization. The next four pages are an index of the file names, which are based on the scientific names and the common names. Color prints of these images are included on the following pages. Instructions for using these images as cross textures and geotypical textures are included in the help system.

Graphics of example projects

The last three pictures in the appendix show some of the basic functionality of Terramodel Visualizer.

Existing ground - Simple terrain models can reveal subtle details when viewed in Terramodel Visualizer. Any project file can be immediately viewed in Terramodel Visualizer to provide a more comprehensive understanding of the project. With minimal configuration of the viewing environment additional details are revealed that might otherwise be lost. (See page xxvii.)

The points in this project file were collected using standard photogrammetric techniques. A single closed set line was created around the lake. A single geotypical texture (water) has been applied to the lake. The points and TIN lines have been included to show how Terramodel Visualizer maintains the full definition of the DTM. A spot type of light has been applied behind the dam to emphasize its limits. As the water in the lake rises, can you spot where it will bypass the dam? This page is a color copy from an original produced with an HP photo printer.

Pipe line - Terramodel Visualizer can visualize multiple DTM layers from projects that can never be seen by the human eye. Terramodel Visualizer can easily visualize any combination of terrain models created within Terramodel. Terramodel Visualizer helps you to understand the complex interaction between existing, proposed and as-built conditions. (See page xxix.)

This project file shows a portion of a pipeline laid on the ocean floor several hundred feet deep. The three DTM layers show the existing ocean floor, the proposed limits of the material used to cover the pipe and the as-built DTM of where the material was placed. A set line was draped along the existing terrain and used in the design. Using the layer and name of the set, pipe properties were assigned to the set in the Terramodel Visualizer environment. Terramodel Visualizer is used to visualize your project from positions the human eye will never travel. This page is a color copy from an original produced with an HP photo printer.

Subdivision lots - Terramodel Visualizer works with your design data to not only better understand the design but to create a finished presentation. (See page xxxi.)

This picture is a combination of several Terramodel Visualizer images created from a single environment showing different aspects of the same project file. These images were combined in Adobe Photoshop and portions of each image removed to reveal how the fished presentation is based on real survey and design objects. This image reveals contour lines, points, the TIN, the set lines of the roadway, the lot lines draped

onto the design, geotypical textures applied to portions of the DTM and trees in the back ground using cross textures. Terramodel Visualizer's unique ability to place lines, trees and textures using Terramodel's point based project file makes your visualization quick, easy and accurate. High resolution images created by Terramodel Visualizer can be edited by programs such as Photoshop to add your company logo and other finishing touches. This page is a color *original* from an HP photo printer.

Tree and Shrub File Names (see page xix.)

File Name	Description	File Name	Description
Row 1		Row 4	
4ABIPIN0.TIF	Spanish	4ELAANG1.TIF	Russian Olive
Fir•4ACALON0.TIF	Golden	4ERYCAF0.TIF	Coral Tree
Wattle•4ACEGLA0.TIF	Rocky Mountain	4EUCCAM0.TIF	Red Gum
Maple•4ACEGRI0.TIF	Paperbark	4EUCCLA0.TIF	Sugar Gum
Maple•4ACEPSE0.TIF	Sycamore	4EUCFIC0.TIF	Red Flower Gum
Maple•4ACERUB0.TIF	Red	4EUCSID0.TIF	Red Iron Bark
Maple•4ACESAC0.TIF	Silver	4FAGSYL0.TIF	European Beech
Maple•4ALBJUL1.TIF	Silk Tree	4FICMAC0.TIF	Moreton Bay Fig
	•4ALEMOL0.TIF	4FRAAME0.TIF	White Ash
	Kukui•4ARAARA0.TIF	4FRAAME1.TIF	White Ash
	Monkey Puzzle		
Row 2		Row 5	
4AREROM0.TIF	Queen Palm	4FRAOXY0.TIF	Raywood Ash
4AREROM1.TIF	Queen Palm	4GINBIL0.TIF	Maidenhair Tree
4ARTALT0.TIF	Breadfruit	4GINBIL2.TIF	Maidenhair Tree
4BETPAP0.TIF	Paper Birch	4GLETRI0.TIF	Sunburst Locust
4BETPAP1.TIF	Paper Birch	4GYMDIO0.TIF	Kentucky Coffee Tree
4BETPEN2.TIF	Euro White Birch	4JUNCHC1.TIF	Blue Column Juniper
4BRAPOP0.TIF	Bottle Tree	4JUNOST0.TIF	Utah Juniper
4CALCIT0.TIF	Red Bottlebrush	4JUNVIR0.TIF	East Red Cedar
4CARGIG0.TIF	Saguero Cactus	4JUNVIR1.TIF	East Red Cedar
4CARGIG1.TIF	Saguero Cactus	4KOEPIP0.TIF	Chinese Flame
Row 3		Row 6	
4CATSPE0.TIF	Western Catalpa	4KOEPIAN0.TIF	Goldenrain Tree
4CEAARB0.TIF	Wild Lilac	4LIQSTY1.TIF	American Sweet Gum
4CEDDEO0.TIF	Deodar Cedar	4LIQSTY2.TIF	American Sweet Gum
4CERSIL0.TIF	Carob Tree	4LIQSTY3.TIF	American Sweet Gum
4CHAObT0.TIF	Hinoki Cypress	4MAGGRA0.TIF	Southern Magnolia
4CHAObT1.TIF	Hinoki Cypress	4MAGSOU0.TIF	Saucer Magnolia
4CORCOL0.TIF	Turkish Hazel	4MORALB0.TIF	White Mulberry
4CORIND0.TIF	Blue Dracaena	4MORALB1.TIF	White Mulberry
4CUPSEM0.TIF	Italian Cypress	4OLMBET0.TIF	Manzanote
4CUPSEM2.TIF	Italian Cypress	4PARACU0.TIF	Jerusalem Thorn

Tree and Shrub File Names (see page xxi.)

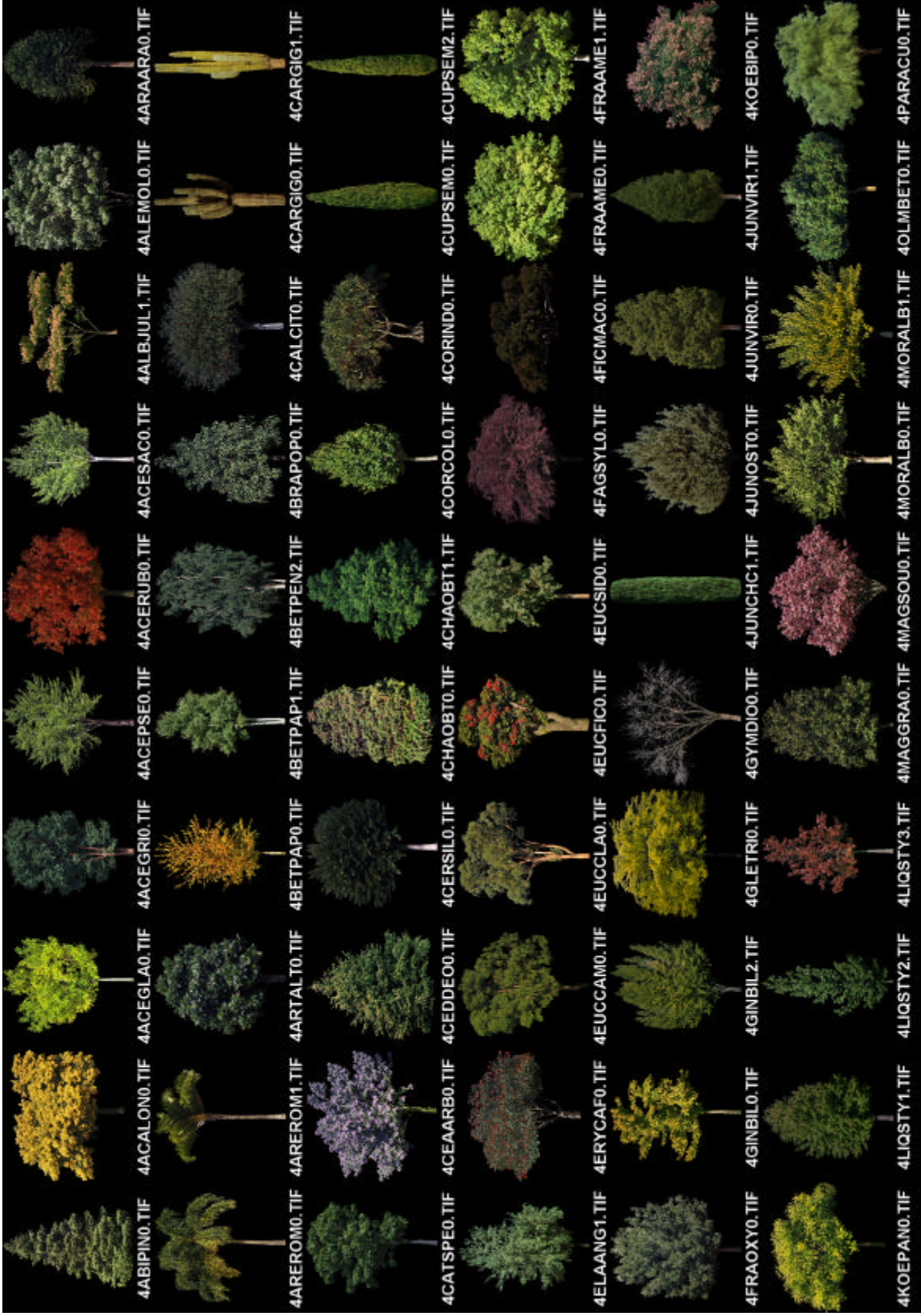
File Name	Description	File Name	Description
Row 1		Row 4	
4PERBOR0.TIF	Redbay Laurel	4SCHTER0.TIF	Brazilian Pepper
4PHOCAN1.TIF	Date Palm	4SEQGIG1.TIF	Giant Sequoia
4PICPUN0.TIF	Colorado Spruce	4SEQSEM0.TIF	Coast Redwood
4PICPUN1.TIF	Colorado Spruce	4SEQSEM1.TIF	Coast Redwood
4PICPUN2.TIF	Colorado Blue Spruce	4SOPJAP0.TIF	Pagoda Tree
4PICPUN3.TIF	Colorado Blue Spruce	4SORAME0.TIF	American Mountain Ash
4PICPUN4.TIF	Colorado Blue Spruce	4TILCOR0.TIF	Little Leaf Linden
4PINCAN0.TIF	Canary Islands Pine	4ULMAME0.TIF	American Elm
4PINCON0.TIF	Lodgepole Pine	4ULMAME2.TIF	American Elm
4PINNIG0.TIF	Black Pine	4ULMPUM0.TIF	Siberian Elm
Row 2		Row 5	
4PINPAR0.TIF	White Pine	4UMBCAL0.TIF	California Laurel
4PINPON0.TIF	Ponderosa Pine	4WASFIL0.TIF	California Fan Palm
4PINTHU0.TIF	Japanese Pine	5ACEMAC0.TIF	Big Leaf Maple
4PISCHI1.TIF	Pistache	5ACEPAL0.TIF	Coral Bark Maple
4PITRHO0.TIF	Pittosporum	5ACEPAL1.TIF	Japanese Maple
4PITUND0.TIF	Victorian Box	5ACESAC0.TIF	Sugar Maple
4PLAOCC0.TIF	American Sycamore	5ARAHET0.TIF	Norfolk Island Pine
4POPANG0.TIF	Cottonwood	5CHAHUM0.TIF	Fan Palm
4POPBAL0.TIF	Balsam Poplar	5CHRLUT2.TIF	Areca Palm
4POPFRE0.TIF	Cottonwood	5CORFLO0.TIF	Dogwood
Row 3		Row 6	
4POPNIG0.TIF	Lombardy Poplar	5CRAAMB0.TIF	Hawthorn
4POPTRE0.TIF	Quaking Aspen	5FAGGRA0.TIF	American Beech
4PRUCER0.TIF	Flowering Plum	5JUNSCO0.TIF	Blue Juniper
4PYRCAB0.TIF	Bradford Pear	5JUNSCO1.TIF	Blue Juniper
4QUEALB0.TIF	White Oak	5JUNSCO2.TIF	Wichita Juniper
4QUESUB0.TIF	Cork Oak	5LIRTUL0.TIF	Tulip Tree
4ROBPSE0.TIF	Black Locust	5MAGSOU0.TIF	Saucer Magnolia
4ROYREG0.TIF	Cuban Palm	5PHOCAN0.TIF	Date Palm
4SALBAB1.TIF	Weeping Willow	5PHOCAN1.TIF	Date Palm
4SCHMOL0.TIF	California Pepper Tree	5PICPUN0.TIF	Colorado Spruce

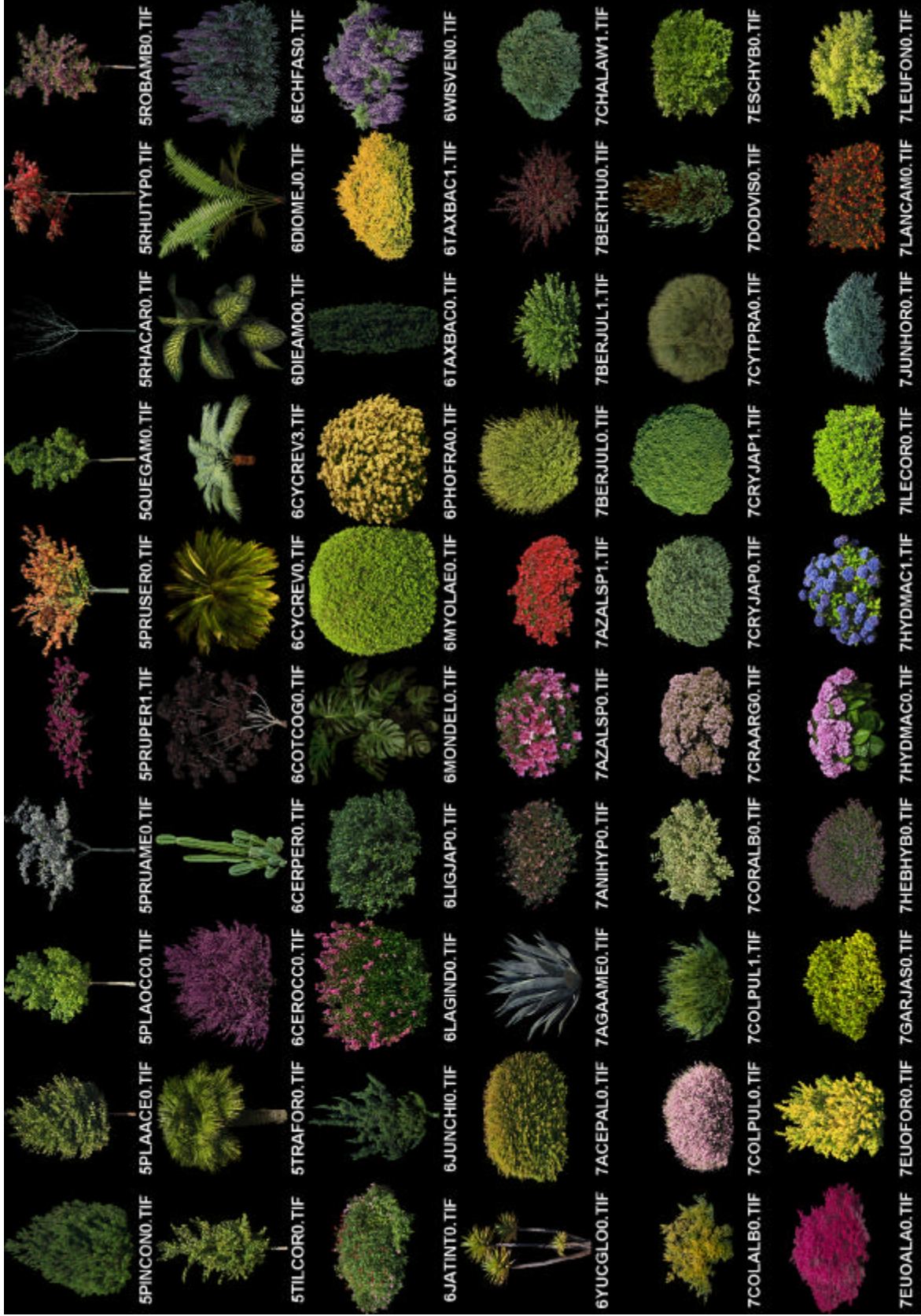
Tree and Shrub File Names (see page xxiii.)

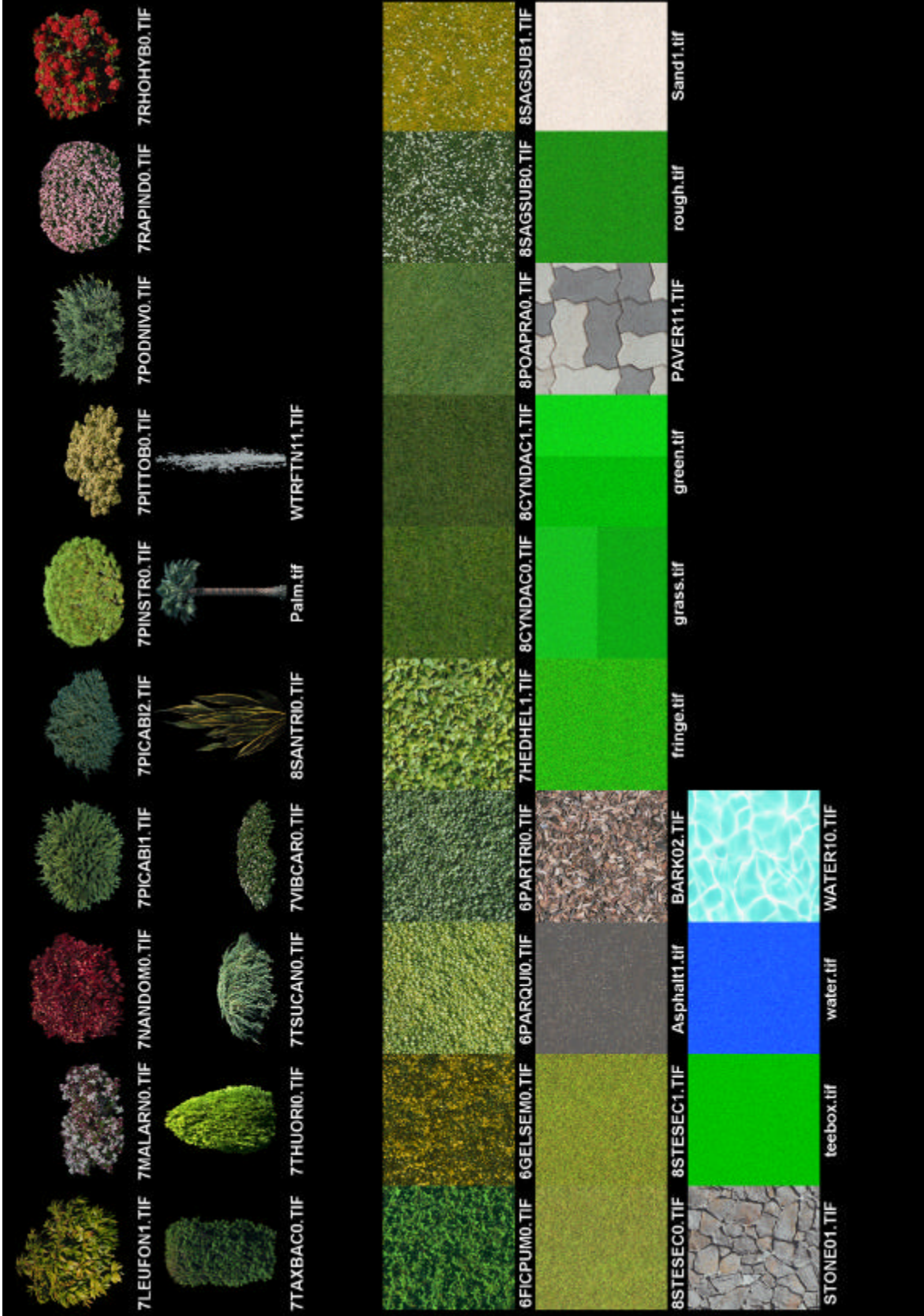
File Name	Description	File Name	Description
Row 1		Row 4	
5PINCON0.TIF	Beach Pine	6YUCGLO0.TIF	Spanish Dagger
5PLAACE0.TIF	London Plane	7ACEPAL0.TIF	Japanese Maple
5PLAOCC0.TIF	American Sycamore	7AGAAME0.TIF	Century Plant
5PRUAME0.TIF	American Plum	7ANIHYP0.TIF	African Mallow
5PRUPER1.TIF	Flowering Peach	7AZALSP0.TIF	Azalea
5PRUSER0.TIF	Flowering Cherry	7AZALSP1.TIF	Azalea
5QUEGAM0.TIF	Gambel Oak	7BERJUL0.TIF	Barberry
5RHACAR0.TIF	Buckthorn	7BERJUL1.TIF	Barberry
5RHUTYP0.TIF	Staghorn Sumac	7BERTHU0.TIF	Red Barberry
5ROBAMB0.TIF	Idaho Locust	7CHALAW1.TIF	Globe Cypress
Row 2		Row 5	
5TILCOR0.TIF	Little Leaf Linden	7COLALB0.TIF	Breath of H
5TRAFOR0.TIF	Windmill Palm	7COLPUL0.TIF	Breath of H
6CEROCC0.TIF	Western Redbud	7COLPUL1.TIF	Breath of H
6CERPER0.TIF	Cereus	7CORALB0.TIF	Dogwood
6COTCOG0.TIF	Smoke Tree	7CRAARG0.TIF	Jade Plant
6CYCREV0.TIF	Sago Palm	7CRYJAP0.TIF	Plume Cedar
6CYCREV3.TIF	Sago Palm	7CRYJAP1.TIF	Cryptomeria
6DIEAMO0.TIF	Dumb Cane	7CYTPRA0.TIF	Broom
6DIOMEJ0.TIF	Jamaican Sago	7DODVIS0.TIF	Hopseed Bush
6ECHFAS0.TIF	Pride of Madeira	7ESCHYB0.TIF	Escallonia
Row 3		Row 6	
6JATINT0.TIF	Peregrina	7EUOALA0.TIF	Burning Bush
6JUNCHI0.TIF	Juniper	7EUOFOR0.TIF	Gold Euonymus
6LAGIND0.TIF	Crape Myrtle	7GARJAS0.TIF	Veitchii Jasmine
6LIGJAP0.TIF	Wax Leaf Privet	7HEBHYP0.TIF	Coed Hebe
6MONDEL0.TIF	Swiss Cheese Plant	7HYDMAC0.TIF	Hydrangea
6MYOLAE0.TIF	Myoporum	7HYDMAC1.TIF	Hydrangea
6PHOFRA0.TIF	Fraser Photina	7ILECOR0.TIF	Chinese Holly
6TAXBAC0.TIF	English Yew	7JUNHOR0.TIF	Blue Juniper
6TAXBAC1.TIF	English Yew	7LANCAM0.TIF	Lantata
6WISVEN0.TIF	Silky Wisteria	7LEUFON0.TIF	Leucothoe

Tree and Shrub File Names (see page xxv.)

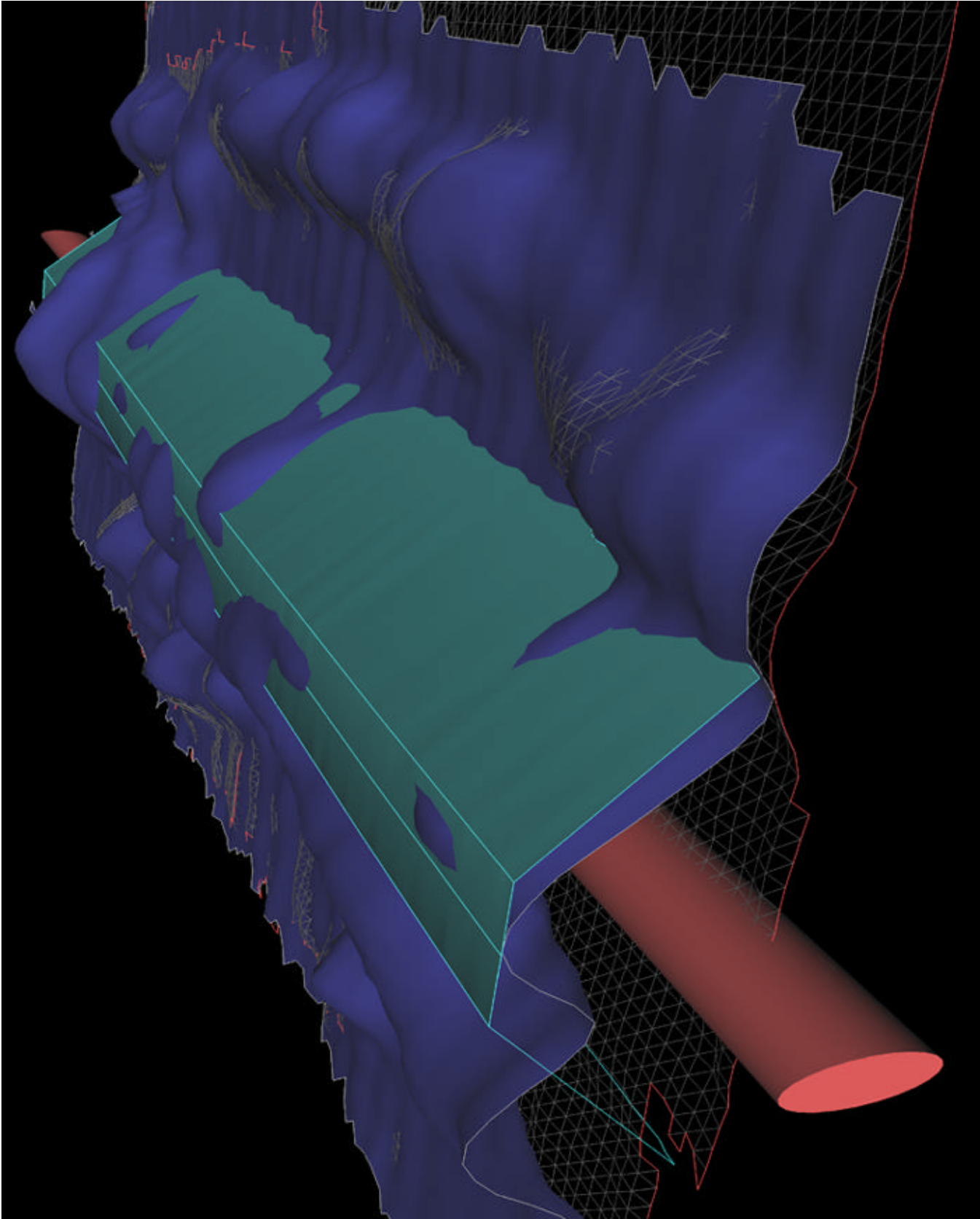
File Name	Description	File Name	Description
Row 1		Row 3	
7LEUFON1.TIF	Leucothoe	6FICPUM0.TIF	Creeping Fig
7MALARN0.TIF	Crabapple	6GELSEM0.TIF	Carolina Jessimine
7NANDOM0.TIF	Bamboo	6PARQUI0.TIF	Virginia Creeper
7PICABI1.TIF	Norway Spruce	6PARTRIO.TIF	Boston Ivy
7PICABI2.TIF	Blue Spruce	7HEDHEL1.TIF	Hahn's Ivy
7PINSTR0.TIF	White Pine	8CYNDAC0.TIF	Bermuda Grass
7PITTOB0.TIF	Pittosporum	8CYNDAC1.TIF	Bermuda Grass
7PODNIV0.TIF	Alpine Totara	8SPOAPRA0.TIF	Kentucky Bluegrass
7RAPIND0.TIF	Hawthorne	8SAGSUB0.TIF	Irish Moss
7RHOHYB0.TIF	Rhododendron	8SAGSUB1.TIF	Scotch Moss
Row 2		Row 4	
7TAXBAC0.TIF	Irish Yew	8STESEC0.TIF	St. Augustine Grass
7THUORIO.TIF	Gold Arborvitae	8STESEC1.TIF	St. Augustine Grass
7TSUCAN0.TIF	Hemlock	Asphalt1.tif	Asphalt
7VIBCAR0.TIF	Spice Viburnum	BARK02.TIF	Tree Chips
8SANTRIO.TIF	Snake Plant	fringe.tif	Fringe Grass
Palm.tif	Palm	grass.tif	Grass Vertical Pattern
WTRFTN11.TIF	Water Fountain	green.tif	Grass Horizontal Pattern
		PAVER11.TIF	Universal
		rough.tif	Rough Grass
		Sand1.tif	Sand
		Row 5	
		STONE01.TIF	Fieldstone
		teebox.tif	Teebox Green
		water.tif	Water
		WATER10.TIF	Pool Water











XXX

