

# Terramodel – Frequently Asked Questions (FAQ's)

## General

### **Q: When entering the name of a prototype file, do I need to include the .pro extension?**

**A:** Yes, when using the box on the System Configuration dialog to set the prototype file, include the full file name including the .pro extension. Otherwise, the program will not be able to find the file.

### **Q: How do I use the On-line Help system?**

**A:** The On-line Help system, arranged according to the menu items across the top of the screen, replaces the Terramodel for DOS Commands manual. While running a command, press the {F1} key on your keyboard to have Terramodel display information about the command. Click the HELP menu on the Menu bar, then click Index, or just press the {F1} key on your keyboard to open the Help system. Once inside the Help system, click on the topic of your choice. Many topics contain highlighted hypertext that, when you click on it, will direct you to more information about the topic or information about related topics.

### **Q: Is there more documentation available about Terramodel?**

**A:** There are several available pieces of documentation available to you that come to you on the program's CD-ROM. There is a Docs directory under Tmodwin on the CD that contains the following executable files.

1) Tmw\_rg.exe.....This is the Reference Guide for Terramodel. It contains detailed information about all of the Terramodel commands.

2) Tmw\_add.exe....This is the Addendum to the User's Guide for version 9.4. It contains detailed information about the new features in version 9.4.

3) Tmw\_ie.exe, Tmw\_ug.exe, Tmw\_tg.exe....These are digital versions of the books shipped with Terramodel. The Instant Expert, the User's Guide, and the Training Guide.

4) Release.wri.....This is the Release notes for the current version.

There is also an Expert directory under Tmodwin on the CD. This directory contains the New Instant Expert books for Terramodel in a digital format.

1) Iegst.exe.....Getting Started with Terramodel.

2) Iegra.exe....Grading Projects.

3) Ieccm.exe....Creating a Contour Map

4) Ierdp.exe....Roadway Data Projects

### **Q: When should I use the command line versus the Pull-down menus?**

**A:** Use the command line when you know the names of the commands you want to use and are more comfortable typing them than picking them from the menus. Use the pull-down menus when you aren't sure of the spelling of the command name or you want to pick commands transparently (while you're still in another command) from the menus. When typing commands from the command line, you can only enter commands transparently by selecting Command line from the SETTINGS menu. You can also add alias command names in the [Alias] section of your TMODWIN.INI file. Terramodel will use these shortcut names instead of the standard command names. Note: You will not see your additions to the [Alias] section until you restart Terramodel. This is true when they are placed using the 'Alias' command from inside Terramodel as well.

### **Q: How do I enter abbreviated command names like I did in the DOS version of Terramodel?**

**A:** You can establish abbreviated commands or "aliases" in the [Alias] section of the TMODWIN.INI file. Use a text editor (or the Windows Notepad) to add abbreviated commands to that section. An ALIAS.ADD file containing alias command names is also supplied and automatically installed when you install your Terramodel disks. To use this file simply paste it into the [Alias] section of the TMODWIN.INI file. There is also a TML called 'Alias' that can be run from inside Terramodel as well. This TML will provide a dialog box for editing the [Alias] section. Again, note that none of these changes will be available until the next time you start Terramodel.

### **Q: How do I edit the TMODWIN.INI and P3SERVER.INI files?**

**A:** Use any text editor to edit the files, which are in ASCII format. Make sure to save the changes in a plain, ASCII format. Or, you can use Window's File Manager and open the directory in which Terramodel for Windows is stored.

Double-click on the icon of the file you want to edit to open it in Window's Notepad. Use Notepad's text editing features to insert or delete text. Always create a backup file before editing any file!

**Q: How do I set my path so that, whenever I begin a new project or open an existing project, I am in the correct directory?**

**A:** With Version 9.4, Terramodel remembers the last directory from which you opened a project, and will automatically bring up the File Open dialog box to that location.

## **AutoCAD Transfer**

**Q: How can I map Terramodel objects' color to AutoCAD's ColorByLayer?**

**A:** You can change the color of Terramodel objects to ColorByLayer by changing the color of objects to color zero (0), which will be ColorByLayer when exported to AutoCAD but will also change the color of objects to be the color of the object's layer in Terramodel. Or, you could change the DXFOPCO.MAP file and map all Terramodel colors to color number 256. This will map colors to ColorByLayer when exported to AutoCAD but will not affect the objects in the Terramodel project. The format of the DXFOPCO.MAP file is as follows:

Terramodel color, AutoCAD color

Example:

0,0

1,256

2,256

3,256...



**Q: When I export from Sheet View, I don't see anything in my .DWG file.**

**A:** With the release of Terramodel 9.5, a change was made in the export of entities from Sheet View. Now, any entities exported from Sheet View are placed in Paper Space in the .DWG file.



**Q: When I import a .DWG to Terramodel, I don't get any points, just Plines.**

**A:** Another change that was made at the time of Terramodel 9.5 is that all linework is imported as Plines whenever possible. Users can still create Sets and Points from these Plines by using the Convert command. With the file being imported as Plines, and then using Convert to create Sets and Points, users now have more control over which entities have points.

**Q: When I map my Terramodel linetypes to AutoCAD linetypes, they come into AutoCAD with the correct name, but they show up as Continuous or Dotted linetypes.**

**A:** When mapping linetypes in DXFOPLT.map file, linetypes on the right side of the file MUST "pre-exist" in Terramodel. This means that the name on the right must already be defined and loaded into Terramodel in order to be defined properly when translating into DXF or DWG formats. Simply add that linetype to the TMODEL.lin file in the TMODWIN directory, then load it into your project under settings - linetype settings or clicking the down arrow next to the current linetype and clicking the "Load" button.

**Q: How do I translate AutoCAD's Paper Space and Model Space in TERRAMODEL and vice versa?**

**A:** When you import a DWG file that has both Model Space entities and Paper Space entities, the Paper Space stuff goes into our Sheet View and the Model Space stuff goes into our Plan View. This does NOT work in the other direction, you MUST export one view at a time and import it into the proper space in AutoCAD. Hint: When exporting back to AutoCAD, use the "View" selection snap, which will grab ALL entities regardless of their visibility. This avoids missing objects which may be turned off or have their visibility off.

**Q: When I bring in an AutoCAD file, it is 12 times larger than it is supposed to be.**

**A:** AutoCAD has Architectural units, so many times the information from AutoCAD needs to be scaled in Terramodel by a factor of 1/12 or .083333333... from an origin of 0,0.

**Q: When I bring my file into AutoCAD, it says it can't find text styles that are in my Terramodel project file.**

**A:** AutoCAD has Styles and Fonts for text entities: When a Terramodel project file is sent to AutoCAD, you will get "style name not found": This will halt AutoCAD: Create a new style name and assign it a font by simply typing

"Style" at an AutoCAD command line, then giving it the name it can't find, assign it a font, import the file again, and move on until it stops again.

**Q: When I send a file to a client, the text and symbols are not imported properly.**

**A:** AutoCAD is looking for the two main font files that are inside of Terramodel (SYMBOL and TMODEL.F): AutoCAD's format for these font files is SHX and are provided for all of Terramodel's standard fonts and located in the TMODEWIN directory. When exporting a file to AutoCAD, make sure you send SYMBOL.shx and TMODEL.F.shx along with any DWG files to ensure that your client can utilize text and symbols that were created in Terramodel: Don't forget that if you use other Terramodel fonts to send the corresponding .shx files along also. When sending the font files to your client, have them copy them into the ACAD\FONTS directory (or wherever their other font files are located).

---

## **Digitizers**

**Q: What is the difference between a "WinTAB" driver and the digitizer drivers supplied with Terramodel?**

**A:** WinTAB Drivers: If your digitizer is supplied with a Wintab Driver from the manufacturer you can install this driver to allow your digitizer puck to act like a mouse in Windows. In Terramodel for Windows the puck will continue to act like a mouse. If you need to use the digitizer board as a digitizer, you can configure the setup option for your tablet to be *Wintab* under File-Config System-Tablet. It is recommended that you have a separate mouse available on your system as well because once the digitizer board is on, the puck will no longer act as a mouse; thus preventing you from accessing any of the pull down commands. Also, please note that Terramodel for Windows comes as both a 16-bit and a 32-bit version as of the release of version 9.4. Make sure that your WinTAB driver matches this. For instance, if the WinTAB driver is listed as for "Windows95" it is probably a 32-bit driver. Terramodel Drivers: If you choose, you can setup your digitizer to function with one of the interface drivers supplied with your Terramodel for Windows program. Again, you will need a separate mouse to access the commands but the digitizer will be able to function as a digitizer once you turn it on using the proper driver for your digitizer under File-Config System-Tablet. The baud rate, parity, data bits and stop bits are determined by the switch settings on your digitizer. It is not important to be concerned with the port settings in Windows as the settings in Terramodel will determine the communication.

For more information on Digitizing, see the "Getting Started" Instant Expert booklet provided on your version 9.4 CD-ROM. It is located in [cd drive]:\tmodwin\expert\iegst.exe

**Q: How do I setup my digitizer once I have determined the switch settings, made sure they match the setup, and chosen the appropriate TERRAMODEL digitizer driver for my digitizer?**

**A:** Under File-Config System-Tablet, choose Setup and make the necessary input for your digitizer, then choose the Menu button and set the number of menus to zero. For the Screen Area, you can set the 'lower left' and 'upper right' coordinates to an area of your digitizer that you will not have a drawing taped onto. It is very important that the 'Screen Area' not overlap the drawing you want to digitize. When the puck enters the Screen Area, it will no longer be working on the registered coordinate system. If Terramodel does not input coordinate values into the screen fields, there is a communication problem with your COM port. After setting the Screen Area, choose the Orient button and perform a two point orientation (for working in Plan View) or a three point orientation (for working in X-Sections) by first choosing point one on the tablet corresponding to the first known point of the item you are digitizing that has been taped to your digitizer, then by choosing the corresponding coordinates in the Terramodel file. You can do this by entering the coordinates, or by using the small 'mouse' icon at the end of the box to go into the project file and graphically pick the coordinates. Follow the same procedure for the other orientation point or points.

Note: After you complete the Orientation process, the program will display a Scale and Angle in the Message Area. Always check these numbers. The scale should be close to the listed scale of the drawing you are digitizing. And the Angle should reflect the direction of North on the drawing on the digitizing board ('0' to the top of the board, '90' to the right, '270' to the left, etc.). By doing the Orientation process you have told Terramodel how to translate movements on the digitizing board into coordinates in Terramodel. The accuracy with which you set this up will affect all of the data that you enter from the digitizer.

**Q: TERRAMODEL does not appear to be accepting input from my digitizer.**

**A:** Confirm that you have oriented to the correct serial (COM) port by using HyperTerminal or some other modem

program by orienting it to receive directly from the port that your digitizer is attached to. When the digitizer is sending data that HyperTerminal is receiving, you should see a steady stream of numbers scrolling on your screen whose values change as you move your puck around. Once you have confirmed communication as well as the port, use that COM port in Terramodel.

---

## *Draft*

### **Q. Is it possible to label a HAL without getting all of the labels for each break in the line?**

**A.** Yes, In the settings for Label HAL, there is a box for "Point Labeling Length". Normally, this setting controls the length of the line that is drawn at each point along the alignment. However, this box can be set to a special value of "-1". This acts as a flag, which tells the program not to label any PI's that don't have curve data.



### **Q. When I attempt to "Label Points with Blocks", the points don't label?**

**A.** Two items to check here....

1st.... What layer are the Points on? If the Points are on Layer '0', then they won't label. Layer '0' is a special layer in Terramodel, and users should never put data on this layer. Points on Layer '0' can not be labeled, and this layer can not be contoured.

2nd.... When you are looking at the "Label Points with Blocks" command bar, what does the first button on the left end of the command bar say? If it reads "Thaw", then all of the labels in the project are frozen, and pressing the button will 'thaw' the labels. The ability to 'freeze' the labels gives users the ability to globally turn off the display of the labels without changing the labeling of the points. The caption on the button tells what the button will do when pressed. So if you want to see your labels, it should say 'Freeze'. If it reads 'Thaw', then the labels are frozen and not visible.

### **Q: How do I change the size of individual point labels?**

**A:** Terramodel supplies a series of sample point label blocks that are available in sizes of .08, .10 and .12 drawing units. To change the size of a point label, use the Settings option of Draft-Label Points, click the down arrow next to the Description list box, and select the point block in a different size (the block sizes are at the beginning of the block names). These blocks are stored in the files name ptlab###.blk in the directory where Terramodel is installed. This system is designed to be flexible to allow users to have point label blocks that meet their needs. You can create your own Point Label blocks (up to 255 of them) that label points the way you want them labeled. A command called "PTLAB" has been supplied to aid you in loading point label blocks, editing them and saving the blocks, or in creating your own blocks from scratch.

---

## *Draw*

### **Q: How can I tell which blocks are available for my project?**

**A:** Version 9.4 has added the concept of a Terramodel Search Path, and a command 'TSP' to allow you to list files available in the search path. The search path begins with the directory from which the project file was loaded, then goes through any directories defined as the TSP in the tmodwin.ini file, and ends in the directory from which the tmodwin.exe file was executed. Typing TSP at the command line will bring up a dialog displaying the directories in this path, and you can set the 'Files' to \*.blk to have the blocks available to you in the TSP listed in the bottom half of this dialog.

### **Q: How do I change the tolerance for drawing sets for arcs (i.e. picking the PC, the RP twice, then the PT) so that Terramodel will draw arcs when the radius lengths differ by only a very small amount?**

**A:** In the TMODWIN.ini file under the [System] heading, there is a setting for SetArcTol. Increase this number to allow Terramodel to draw not so perfect arcs. Remember that Terramodel will compute the arcs to the nth decimal place so the legs WILL remain unequal, but it will allow you to draw the arc for display purposes. Also, you should not reduce this tolerance below what you deem acceptable limits (.005 to .01 max. if working to two decimal places).

---

## **DTM's**

### **Q. When I'm using the XSheet command to create cross-section Sheets, how to I get cross-sections for multiple DTM's?**

A. When a user presses OK on the command bar to execute the XSheet command, a layer selection box entitled "XSheet Layer Selection" appears. It is possible to select multiple layers from this dialog using standard Windows mouse conventions. By holding the 'CTRL' key down while selecting with the mouse, multiple layers can be selected. And by holding the 'SHIFT' key down, a range of layers can be selected by picking the first and then the last layer of the desired range. When the user presses OK from this dialog, the software will attempt to cross-section all of the layers that are highlighted in this dialog.



### **Q. My points won't contour.**

A. What layer are the points on? If they are on layer '0', then they won't contour. Layer '0' is a special layer in Terramodel (see below), and users should never place their data on this layer. Layer '0' will not contour in Terramodel, and also, points on that layer cannot be labeled.



### **Q. When the program is linking a surface, I see a message about duplicate points being moved to layer '0'. What's happening?**

A. While Terramodel is linking a DTM layer, it checks for points with identical Northing and Easting coordinates. When it finds such duplicate points, it moves one of the points to layer '0', and keeps the other point on the DTM layer for use. If the two points also have identical elevations, it checks any sets which are using these points, and adjust the sets such that any that were connected to the point that is being moved to Layer '0' are reconnected to the point which remains on the DTM layer. This is why users should never put good data on layer '0'. The program will move duplicate points to this layer, and users will probably want to follow this up by periodically deleting all entities on layer '0'.

### **Q: What are some things I can look into when the Design command tells me it is "Unable to Design"?**

A: Some things to check with the Design command:

1. There needs to be a set around the Design layer.
2. The set defining the edge line must be on the Design layer.
3. ALL points that have an elevation on the Design layer must be INSIDE the set.
4. ALL points defining the edge line set MUST have an elevation.
5. Make sure that the design does not extend beyond the limits of the surface that it is designing to.
6. There may be points on the edge line set that are not part of the set. If this is the case the set MUST be altered to include these points.

One way to check for such points is to: Inquire-List-Points (NearLine selection - offset of 0.01 - and select the edge line set), then Remove from this selection (Offline - select the edge line set) . What ever is left in the selection is the points within 0.01 of the set but NOT part of the set.

### **Q: I change my design slopes from 2:1 to 3:1 using Settings-Design Settings, but when I re-design, it still designs at a 2:1.**

A: Changes to the DesignSet settings only affect sets that are created after the settings are changed. To change the

---

design slope of existing sets, use the Edit object command, Slopes option.

## ***Edit***

### **Q: When I try to edit a polyline, what does the "355:1" mean?**

A: The first number, in this case 355, is the record number in the database. The second number is the polyline's control point, or vertex, at which Terramodel's internal pointer is directed. If you highlight the "Cp:" box and select a vertex, the second number will change to that of the selected vertex. When the focus is in the Cp: control, you can use the up and down arrow keys to advance to other vertices along the polyline.

---

## ***Hardware Lock***

### **IMPORTANT**

**Do not run any type of Lap Link or Backup device through your Terramodel for Windows lock. This will cause the lock to go bad and you will not be able to access Terramodel until we can send you a replacement.**

**When experiencing Lock problems try the following:**

1. Remove the lock and any additional devices from your parallel port (additional locks or printer or plotters).
2. Reboot your system with your lock off.
3. Replace just the Terramodel lock onto your parallel port.
4. Try to access your Terramodel for Windows program.
5. If the lock is not still not recognized, try the lock on another computer system if possible. This might help determine if it is a problem with the lock or with the computer.
6. The 'mode' of the printer port can sometimes cause problems. Look at this particularly when the lock fails to work on a new computer. To check the mode, you usually have to go into the BIOS settings on system boot-up. Look for a section on parallel ports. If the mode is set to 'EPP' or 'ECP' try setting the mode to something like "Standard", "Normal", "Conventional". **IMPORTANT:** Write down the changes that you make in case you have to undo it.
7. Sometimes loading the hardware lock drivers will help a Windows 95 machine. The latest drivers can be obtained from <http://www.rainbow.com>. We also have a set in the "Locks" directory on our CD-ROM.
8. If you are still having trouble-getting Terramodel to recognize your lock, call our technical support staff at 1-800-235-4972 for assistance.



## ***Hydrology***

**Q. When building a Stage-Storage table, can I assign an Area of zero to an Elevation?**

**A.** No. Instead of zero, you need to use a small number. An area of 0.1 will work instead. In general, if the Hydrology seems to be routing poorly, or if the Storage volumes don't seem to be correct, this is usually a good item to check.



**Q. When I add storms to the Rainfall table, the Rainfall Reports command doesn't report these.**

**A.** There is a bug in the Rainfall Reports tml that reports rainfall totals used in SCS hydrographs. The bug is only in the report generation, the correct storms are being used to generate the Hydrographs. There is a revised TML in the TML section available to Active Members. This revised TML correctly displays the data for storms selected by the user under Hydro Defaults.



**Q. How do I create my own Rainfall file?**

**A.** The best way to do this is to make a copy of the atlanta.rnf file that is supplied with the software and edit that file. This should provide a file in the format that the program is looking for. A couple of other pointers in creating this file....

- The program ignores all lines in the file that begin with a '#' sign. It also ignores all empty (zero length) lines. Make sure you don't change this format.
- The first line in the file that is neither empty nor which doesn't begin with a '#' sign must have the line which defines the available storms.
- Don't have spaces in the available storms line after the last number.
- After the available storms line, the program reads the I-D-F information used with the Rational Method. This is followed by one line that reads "TOTAL". The rainfall total data used with the SCS Method follows this line.
- You can have up to 11 storm periods defined in this file.

## ***Image Manager***

**Q: Why isn't my image visible while using the Image Manager module?**

**A:** Follow these guidelines when attaching an image file:

1. When you enter the name of the image file in the Image Manager dialog box, make sure you type in the complete path and extension.
2. Verify that the image file is less than 32 megabytes in size. This is a limitation in our 16-bit versions. The 32-bit versions removed this limitation.
3. Make sure that the coordinates for the upper right and lower left corners of the image are entered.
4. Use Zoom to view the coordinates of the image corners to be sure you are actually looking at the correct area.

The image will not plot if the plot is rotated by an amount other than 0°, 90°, 180°, or 270°, or if the image is being viewed through a dynaview. This will be enhanced in a future version. You must use your Windows driver to plot an image using Image Manager.

---

## *Inquire*

**Q: When I try to generate a station & offset report (COGO Reports-Sta, Offset or Sta, Offset, Elev. option) for a selected alignment, all I get is the coordinates.**

**A:** Before generating the report, specify the alignment to be used for station and offset information as the active alignment using Settings-Active Alignment or type Active at the command line.

---

## *Installation*

**Q: How do I install from the CD?**

**A:** Simply place the CD in the CD Drive and go to your Start button in Windows 95 or NT 4.0 (use File-Run from your Program Manager in Windows 3.1), browse on drive D (or your CD drive letter) and look in the TMODWIN directory. For version 9.4 only, under this directory you will find two directories named tmw16 and tmw32. These are for the 16-bit and 32-bit versions of the program. Once you are in the proper directory, look for a file called SETUP.exe. Double click on it, which will fill in, the Run line and click "OK" to run the setup. Follow the instructions, choose the installation type, and the directory to install to, then sit back and wait for the installation to finish. If you have a previously installed version of Terramodel on your system, the installation should ask you some questions about overwriting files near the end of the installation. Be careful to read which files it will want to overwrite before answering "OK". If you have a previously installed version, you may want to make a backup copy of your TMODWIN.ini and P3SERVER.ini files along with any other files that you have edited. If you run Windows 3.1x, you will need to first install Win 32s by running SETUP from the TMODWIN\WIN32S\DISK1 directory on the CD. If you are running Windows NT, you will need to run INSTALL.bat from the TMODWIN\W32\_LOCK directory on your CD.

**Q: I installed my software on Windows NT, but am getting a message that says "No Connection to DTM Server".**

**A:** More likely than not you did not install the "Lock Software" by running the INSTALL.bat program from the TMODWIN\W32\_LOCK directory on the CD. Software on a WindowsNT system cannot directly access the port without a driver. So installing the driver is mandatory on WindowsNT systems. If you have installed the lock software and are still getting the message, try rebooting your NT. If that still does not work, call our technical support team at 1-800-235-4972.

**Q: What if I do not have a CD drive in my computer?**

**A:** There is a batch file called makedisk.bat on the CD-ROM. Running this batch file will copy the Terramodel installation program onto a stack of floppy disks. Of course, in order to run this, you will have to have access to a computer with a CD-ROM and a stack of 3.5" disks. Spectra Precision Software no longer supplies Terramodel on

---

3.5" disks.

## ***Metric Conversion***

### **Q: How do I create a Metric file from a file that was created in English units?**

**A:** Changing the unit settings does not change the data in the file. If the file was originally created in English units, the data must be physically scaled using Modify-Scale and applying the meters to feet conversion factor of 0.304800609600 from an origin of 0,0. To scale the elevations, use the FACTZ TML by typing "FACTZ" at a command prompt. Don't forget to change the data and display settings to Metric under the Settings-Unit Settings. All of these settings will be saved with the project file.

---

## ***Micro Station Transfer***

### **Q: Why am I getting a "Import of MicroStation File Failed" message?**

**A:** One cause of this has been a file set to be "Read Only". This has been particularly true of files distributed via CD-ROM. You can check this property and change it by Right-Clicking on the file in Windows Explorer and selecting 'Properties' from the menu that appears. There is a Check Box to mark a file as Read Only.

---

## ***PIPE Module***

### **Q: I am having trouble getting the Pipe-Draw Pipe to draw the pipes the correct thickness.**

**A:** In most cases the pipe is drawn at a 0" diameter. Our development staff believes they have determined the cause for this and you will find a replacement Pipe Draw TML available on our Web page or you can contact one of our Technical Support Engineers.

---

## ***Printing and Plotting FAQ's***

### **Q: How can I speed up the plotting time of my drawing?**

**A:** If your plotter is HPGL- or HPGL2-compatible, use the direct drivers supplied by Terramodel. In many cases, the plotting time is controlled by the plotter driver you are using. If you are using a Windows Printer driver, make sure you have selected the one most compatible with your plotter. If you are using a driver other than the drivers furnished by Windows, verify through the manufacturer that it is the latest driver for your particular plotter.

### **Q: How do I set up a plotter in Terramodel?**

**A:** Try these few tips.

1. *Check your Current Penmap.* This file is located under FILE – PLOT SETUP – PLOT PARAMETERS. In here you assign a pen number to a specific screen color.
2. *Set up your Pen Carousel.* This file located under FILE – PLOT SETUP – CAROUSEL SETUP. In this file you can assign the width and shade of your pens. This applies only if your plotter is set to allow the software to handle pen sizing.
3. *Configure your plotter.* Under FILE – PLOT SETUP – PLOTTER SETUP you will find the area to configure a plotter.
  - a. Choose a name for your plotter.
  - b. Edit the configuration.
  - c. Choose a driver, model and port for you plotter.
  - d. Set your sheet size and margin allowance.
  - e. Choose a default pen carousel.

All this information is saved to the TMODWIN.ini file.

### **Q: Can I find more information about HP Printers?**

HP DeskJet Printing Solutions:

Here are some links to Issues and Solutions on HP's Electronic Support Center. We also have a link to the entrance to the HP Electronic Support Center for convenience. [HP DeskJet Series Printers -- How to Troubleshoot Cable / Communication Problems Between the PC and the Printer](#)  
[HP DeskJet Series Printers -- Disabling Bi-directional Communication](#)

## ***Text***

**Q: How can I force my labels to read right side up?**

**A:** For each style you are using when you label, select Legible for the style orientation in the StyleSet command (SETTINGS/ Text Style Settings). This will allow the text to orient itself to the bottom or right side based on the flip over angle established under Settings-Draft Settings and appear right side up, even when viewed through a dynaview.

**Q: How do I create new text styles in Terramodel?**

**A:** Text style names are "bookkeeping" names which store the font to use, the size of the type, the aspect ratio, slant, legibility, and other information about how a particular piece of text is supposed to look. Create new text styles under Settings-Text Style Settings. Create a new name, choose a font and set all other parameters associated with that style of text. This name is saved and can be referred to throughout the project. Hint: Set up standard text styles which you use all of the time in your prototype file.

**Q: Can I create new fonts from SHP files I get from other sources or from SHX files I get from AutoCAD?**

**A:** Terramodel will only recognize fonts in the Terramodel font format that have an FNT file extension. You can convert shape files (.SHP) into Terramodel font files using the FONTC file converter by typing FONTC (space)<FONT\_NAME.SHP at a DOS prompt from the TMODWIN directory (the directory in which the FONTC.EXE file and the shape files are stored). To create an FNT file from an AutoCAD SHX file, use SHX2SHP (space)<FONT\_NAME.SHX at a DOS prompt from the TMODWIN directory (the directory in which the SHX2SHP.EXE file and the AutoCAD font files are stored).

**Q: When I type a line of text that contains more than 255 characters into the Draw-Text-Text dialog box and place it, why is the text truncated?**

**A:** Text created by the Text command (Draw-Text-Text) is really a single text string containing carriage returns. Consequently, any text entered is truncated to the maximum available length of 255 characters. As an alternative, enter the text into a text editor and save it in a file, then insert it using the Draw-Text-From an ASCII file (TxtIn).