The ProtoTRAK® DXF Converter
Operating Manual

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DXF for Mills and Lathes
Compatible with offline and
ProtoTRAK Control models:
VM, VL, SM, SL, SMX, SLX, and PMX

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1.0 Introduction

Thank you for your purchase of the ProtoTRAK DXF Converter software. In order to avoid repetition, this manual assumes you have basic proficiency with programming a ProtoTRAK CNC.

With the ProtoTRAK DXF converter, you may quickly and easily create ProtoTRAK part programs using the data stored in a DXF or DWG file. The ProtoTRAK DXF Converter is the best value on the market, giving you advanced features, such as layering, chaining and automatic gap closing that are normally found only on products costing much more.

With the ProtoTRAK DXF Converter, you are in complete control of the creation of your 2D or 2 ½ D part program. You pick the order of events and the direction you want to machine in an easy, interactive process. The coordinates are loaded into to the part program in the way you direct, leaving you the minimal task of adding the non-dimensional information in the prompted ProtoTRAK format.

1.1 DXF Versions

There is a version of DXF for use in creating 2 or 2 1/2 axis part programs for Milling and a different version for creating part programs for turning.

The part numbers are:
CONV-DXF   DXF converter for mills
CONV-DXFL   DXF converter for lathes

The lathe and milling versions are sold separately. If you want both of them, you may purchase one at the additional seat pricing as long as you order them both at the same time. See your Southwestern Industries Sales Representative.

1.2 DXF Defined

DXF stands for Drawing Exchange File. It is an invention of AutoCAD that has become the most common format in the industry, used by most CAD (Computer Aided Drafting) programs as an alternative file extension.

As well as converting DXF files, this software also converts drawings with the DWG (Drawing) file extension. DWG is the file extension currently used as the default for AutoCAD.

1.3 Mouse Required

You may run the DXF Converter on either the ProtoTRAK CNC or the ProtoTRAK Offline system (depending on for which system you bought it). If you run the converter on the ProtoTRAK CNC, you must have a mouse (or other pointing device) plugged into the mouse port or USB port, depending on the model of control. For ProtoTRAK controls that have USB drives available, a USB mouse should be recognized immediately after plugging in. For the models that have the PS2 mouse port, if you didn't have the mouse plugged in at the time the ProtoTRAK was turned on, you must turn off the ProtoTRAK, plug in the mouse and then turn the system back on.

When running the DXF Converter on the ProtoTRAK CNC, a keyboard may be used but it is not necessary. You can use the front panel keys on the control pendant to input any data necessary throughout all the operations of the DXF Converter.

1.4 Compatibility

Although the DXF format has become an industry standard, compatibility issues remain. The following may help clarify some issues with regard to the DXF Converter.

- This version of the DXF Converter is compatible with all current variations of the DXF file format. However, as CAD/CAM software progresses and come out with newer features, compatibility issues may occur. If you are having trouble opening a particular DXF file, or there are objects or entities that are not recognized, try saving the DXF file as an older format such as 14 or 2000. This usually saves the DXF file as a simpler format, but the drawing itself should remain intact.

- The DWG file format is supported up to version 2006. For versions 2007 and greater, you may need to save as an older file format in order for our converter to be able to recognize. Autodesk has a free software download on their website that can convert DWG files from one version to another. Please visit http://www.autodesk.com

- The DXF and DWG formats were invented by AutoCAD. Other CAD systems that use this format as an alternative have made their products compatible to the AutoCAD standard. Therefore, files created with this product should also work with this converter, as long as those files are saved with the DXF or DWG extension.

- The ProtoTRAK DXF Converter may be used with 2D drawings, it is not intended for 3D or Solid model drawings.

Virus Protection:
As a device, ProtoTRAK CNCs are not generally susceptible to viral infections. The part "programs" they run are non-executable text files. You can further assure protection by avoiding e-mail programs and web browser programs loaded onto the ProtoTRAK and by using a router with a firewall. An anti-virus program is not necessary since the virus risk is low, and is not recommended because the background tasks may cause damage by interfering with ProtoTRAK real-time operation.

Microsoft Updates:
Please note that the ProtoTRAK control is not designed to be automatically downloading updates from Microsoft, or for that matter any 3rd party software. There is only a limited amount of free space on the compact flash card, and downloading files to it until it is full may result in the ProtoTRAK to stop working properly.
Also note, that administrators must exclude the ProtoTRAK from programs such as Windows Server Update Service, where the software will automatically try to download updates to the control.

2.0 Installing and Activating DXF

The DXF Converter is optional software that must be purchased from Southwestern Industries. If you purchased this option with your machine or control, the software should already be installed and activated. If this is the case, you may skip the rest of this section.

You can tell if your converter is activated by looking at the data in Service Code 318 (Service Codes are found in the Set Up Mode). Service code 318 will display a list of all available converters. Active converters are displayed in black letters; inactive converters are displayed in gray letters. If the DXF Converter is not on the list at all, call the Southwestern Industries' Service Department.

If you purchased your DXF Converter separately from your machine or Offline System, you may have to install software first and then activate the option.

Note: as of January 1, 2005, Offline Software comes with separate programs. One for the ProtoTRAK VM, VL, SM and SL and one for the ProtoTRAK SMX and SLX. It is necessary to install the DXF Converter on both of the programs in two separate processes if you want to use the DXF converter with both programs.

2.1   Installing the DXF Software

For ProtoTRAK Controls: For the ProtoTRAK VM, VL, SM and SL, version 3.02 or higher is required for the installation of the DXF software. You may visit our web site at www.southwesternindustries.com and go to "Support", and then "Software Downloads" for the latest version of software. All ProtoTRAK SMX, SLX, and PMX versions will run with the DXF software.

When installing the DXF software, make sure to load in the correct version according to your control. For the ProtoTRAK SM, VM, SL and VL, use the disk with the part number # 23519. For the SMX and SLX, use the disk with part number # 23519-1. The ProtoTRAK PMX should already have the DXF software installed in the control.

For ProtoTRAK SM, VM, SL, VL, SMX, or SLX:
To install, place the correct disk into the floppy drive. Press the SYS or Windows key, and then press the Update Options softkey to begin the process of loading the new software

For ProtoTRAK Offline (all controls):   All current ProtoTRAK offline versions now contain the DXF software included in the installation (activation will still be required). You can download the latest version at www.southwesternindustries.com and not need to install any addition software.

To download the DXF software via a USB flash drive, or to obtain the latest version of any DXF software version, visit our website at www.southwesternindustries.com, go to the Software Downloads section and follow the instructions in order to download the latest software to your ProtoTRAK control.

2.2   Activating DXF

After the software is loaded, you must activate the key (our way of controlling the use of the software) with the following procedure:

1. On your ProtoTRAK control, go to Service Code 318 Activate Converter (Service Codes are found in the Set Up Mode). Here you should see all active and inactive converters. Inactive converters will be listed in Gray. If the DXF Converter is not grayed out (i.e., it is black) it is already activated. Skip the rest of this section and go to 3.0 Loading a DXF part program.

2. Data Forward to the grayed out DXF Converter option and press Install.

3. A different screen will appear on which you will see a Hardware Key serial number, a Converter Serial and a phone number to contact SWI Customer Support. You will also see a blank field labeled "Activation Password". If you already have your activation password, enter that in now and press Inc Set or Abs Set.

4. If you do not already have the activation password, write down the Hardware Key and Converter serial numbers shown on the screen and call SWI Customer Support. After taking the serial numbers, SWI Customer Support will give you your password. Please note, this option is not free. If you have not placed an order for the DXF Converter, you will need to do so before we can give you the activation password.

5. After the password is entered, the DXF Converter is ready for use. Simply restart the ProtoTRAK system so that it recognizes the new software. For the ProtoTRAK CNC on your machine, do the shut down procedure and then turn the ProtoTRAK back on to allow it to reboot. For a ProtoTRAK Offline, it will not be necessary to restart the computer. Simply close the ProtoTRAK Offline application and then start it again.

3.0 Using the DXF Converter for Mills

Using the DXF Converter is a matter of a few simple steps that will be described in this section.

Please set the ProtoTRAK CNC or Offline to the desired 2-axis or 3-axis mode before entering the DXF Converter. These settings cannot be changed once the converter process has started.

Your ProtoTRAK software comes with a sample DXF part program you can use to follow along with these instructions. This part program is named BALLPARK.DXF and it is located on the C drive in the ProtoTRAK CNC in the directory called "SWI Test Programs", within the directory where the main software is installed (PT4, PT4SX, or PT7). For the Offline program, the part program is in the drive you specified when you installed the ProtoTRAK Offline on your computer.

3.1 Opening a DXF Part Program

The DXF Converter is launched automatically when you open a file with the DXF or DWG extension.

When you select a file with an extension of .DXF or .DWG, the Open As field automatically changes to ACAD DXF. Press Open File and the DXF Converter will begin to process the file.

Note: *You can use Mode (Alt + M in Offline) to exit DXF and return to normal ProtoTRAK operation at any time during the DXF conversion process. This will discard any information generated during the DXF conversion session.*

3.2 Choose the Layers That You Want to be Displayed

CAD files are made for printing, not machining. Therefore, DXF and DWG files usually contain information that is good to have on a print, but not useful for machining. Examples include borders, writing and dimension lines. To make the DXF conversion convenient, it is helpful to eliminate those unneeded elements of the drawing from view.

The initial screen of the DXF converter has all the drawing's layers visible. This step allows you to Hide unwanted information so you may focus on what you want to machine. See figures below.
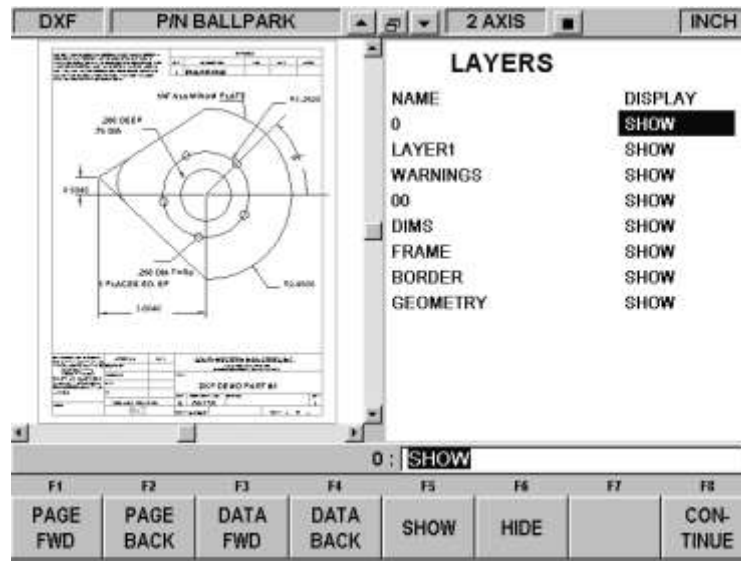

FIGURE 3.2.1 The first screen you see when the DXF Converter launches. Notice that the entire drawing, including borders and dimensions, is shown on this first view.
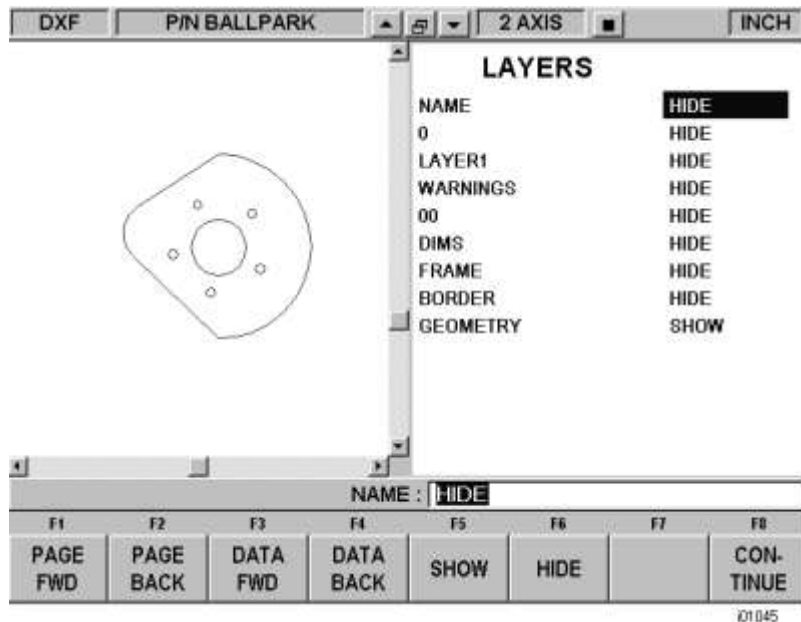
FIGURE 3.2.2 Through a process of elimination, we have isolated the view of what we want to machine.

To choose a layer that you would like to Hide use the Data Forward key or simply click on the layer name with the mouse. You can go between Show and Hide as often as you like as you choose which layers to eliminate from view.

In this first screen, there are some new buttons that don't appear elsewhere on the ProtoTRAK.

Items unique to the DXF Converter appear in the status line at the top:

Background Selection Button. Click on this to toggle between a white and black background. You should use the background that allows you to see the layers on the drawing. So that they remain visible, the DXF Converter will automatically change the color of white lines to black or black lines to white depending on the background you choose. Colors used in the drawing that are other than black or white will stay the same regardless of background.

Zoom In, Zoom Out and Auto-Fit Drawing Buttons.

Press Continue when only the layer(s) you wish to remain are shown. When you leave this screen, you may return to it by pressing the Back key (Alt + B on the ProtoTRAK Offline). However, any additional data you have added will be lost.

3.3 The Gap Tolerance

The ProtoTRAK makes it easy to program pockets and profiles and the DXF Converter makes it even easier by providing the dimensions. One potential problem in this process is that a CAD drawing is made to be looked at, not machined. DXF and DWG files may contain lines or arcs that look connected on a print, but really have tiny gaps between them. The gap tolerance of the DXF Converter is an elegant solution to this problem. You enter a single value – a tolerance – and the DXF Converter automatically closes the gaps that are within that value while you are programming a Pocket or Profile. If you don't want this done automatically, you aren't forced to

apply the tolerance. The DXF Converter will prompt you when it encounters a gap and let you decide whether to close it.

3.3.1 Gap Tolerance Applied Automatically

As soon as you decide your layers and select Continue, you will be asked about the Gap tolerance. If you want to use the 0.005" or 0.15mm gap tolerance in the status line, simply choose YES. You may also input a different value for the gap tolerance and then choose YES. The DXF Converter will refer to this tolerance and will automatically stretch lines an amount that is within the tolerance in order to make them connect. If a gap is encountered that exceeds this tolerance, you will be prompted with a choice to close the gap that is encountered.

If you choose NO, the DXF Converter will prompt you every time the gap is encountered while programming a profile or pocket.

3.3.2 Closing Gaps As They Are Identified During Programming

Once you begin the programming of a pocket or profile, you will be prompted to resolve a gap between lines and arcs that are to be connected when:

1. You input a gap tolerance but the gap that is encountered exceeds the tolerance you entered.

2. You chose NO to the gap tolerance.

In either case, when a gap is encountered, the DXF Converter will display a message like the following:

FIGURE 3.3.2 A gap is encountered that exceeds the gap tolerance.

- If YES is chosen, the displayed gap value will only be used to close the current gap. The original gap tolerance value (if any) will be retained for checking all other gaps encountered.

- If NO is chosen, the gap will not be closed. The original gap tolerance value will be retained and drawing segment will remain unselected.

- If Apply All is chosen, the gap will be closed and this new gap value will replace the previous one as the new gap tolerance value. You will not be prompted again unless a gap outside this new tolerance is encountered.

3.3.3 Gap Tolerance When Chaining

When the chaining capability is used in profile programming (chaining is described in Section 3.5.4 below) the gap tolerance is applied. When a gap that exceeds the gap tolerance is encountered in the chaining process, the DXF Converter will assume it has reached the end of the geometry to be chained. In order to have the chaining continue, simply click on the next geometry to be connected. You will then get the error message shown in figure 3.3.2 with the choice of changing the gap tolerance with the Apply All command.

3.4   Pick a Zero Absolute Reference

The drawing file you converted has its own reference point, but that may not be the most convenient one for programming. The screen you encounter after setting the gap tolerance will allow you to select your own Absolute zero reference. The DXF Converter provides three choices for choosing the Absolute zero reference point that are described below.

3.4.1 Choice A: An Intersection

This allows you to use intersecting points, like corners, as the absolute zero reference. The intersecting points can be taken from two lines, two arcs, or a combination of a line and an arc. Circles cannot be used with this choice.

To make an intersection the absolute reference, select choice A, then click on the two pieces of geometry that will intersect at the point you want to use. After you click on the second piece of geometry, a red mark will appear to show you where the absolute zero is now set.

This choice also allows the use of the intersection of two lines that meet off of the given geometry. If this is the case, you will not be able to see the red mark where the new absolute zero reference is set.

3.4.2 Choice B: An End Point

This allows you to the use an end point of a line or an arc. Click on a line or arc close to the end point to be used as the absolute zero reference. Circles cannot be used with this choice.

3.4.3 Choice C: An Arc Center

This allows you to use the center of an arc or circle as the absolute reference point by simply clicking on an arc or circle.

3.4.4 Continue: Use the Drawing's Absolute Zero

If CONTINUE is pressed without selecting an absolute zero reference point with one of the methods above, then the reference point given in the drawing will be used. This can be helpful if the CAD programmer used a more appropriate absolute zero reference than one that can be created with the given choices. However, in most cases, when CAD drawings are created, the absolute zero reference is usually in one of the corners of the drawing which does not make the best ABS 0 reference point for machining purposes.

Note: *The back key (Alt + B) can be used to return to this screen from any subsequent screens, however, programmed information will be lost.*

3.5   Editing the Drawing

Sometimes the lines of the drawing aren't exactly what you need to machine. For example, you may want to add a ramp at the beginning or end of a profile. You can easily add lines to the print using the Add Line and New Point feature. Once these lines are added, they may be programmed in exactly the same way as the other features of your part.

Add Line: Creates a line between existing beginning or end points of a line or arc. Click on one line or arc at a place nearest to the point to want to use. A red X will confirm the point that is selected. Next, click on another line or arc. The DXF will add a new line between these points you designated. To remove the line immediately after it is drawn, click the Undo button.

New Point: Creates a line between an existing beginning or end point on the drawing, and a new point you establish by clicking the mouse. Click on one line or arc at a place nearest to the point to want to use. A red X will confirm the point that is selected. Next, click wherever you want the line to end. The DXF will add a new line between these points you designated. To remove the line immediately after it is drawn, click the Undo button.

3.6   Converting Geometry into a ProtoTRAK Part Program
Now that the ProtoTRAK has the geometry and absolute reference for the part, programming is just a matter of selecting the events. With the ProtoTRAK DXF Converter, you have total control over how the part is made - just like regular ProtoTRAK programming.

The basic steps are:
1. Pick the event type.
2. Pick the geometry on the drawing. This will load the dimensions from the drawing into the event type you choose.
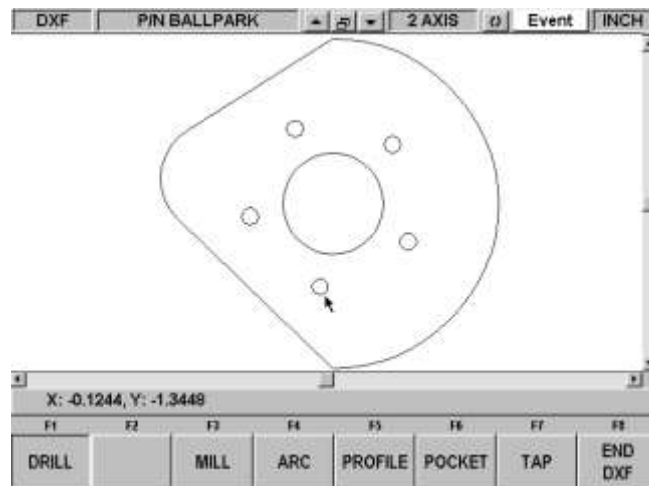3. Fill in the rest of the prompts to complete the event.

FIGURE 3.6 The Drill event has been selected and now it is a matter of clicking on each circle to load in its geometry.

3.6.1 Event Types

Drill. Allows the conversion of any circle into a Drill event. All the same sized drills should be picked at the same time.

Mill. Allows the conversion of any straight line into a Mill event. Click on the end of the line where you want to establish the beginning point (X Beg, Y Beg).

Arc. Allows the conversion of any arc in the drawing into an Arc event. Click on the end on the arc where you want to establish the beginning point (X Beg, Y Beg).

Profile. Allows the conversion of any connecting lines and arcs into a Profile event. When selecting a profile event, the first line clicked determines where the profile starts and the second line clicked determines the direction of the profile. Each line or arc segment in the profile must be connected to each other but the profile may be open-ended.

Pocket. Allows the conversion of any connecting lines and arcs into a Pocket event. When selecting a pocket event, the first two lines clicked determine the direction of the Pocket (Clockwise or Counterclockwise). Pocket segment geometry must be connected to each other and must be closed – it needs to end where it began.

Tap. Allows the conversion of any circle into a Tap event.

3.6.2 Functions in the Status Line

Zoom In. Click on this button to zoom in on the view of the drawing. Zooms in by 25% of the drawing's current size each time the button is clicked.

Zoom Out. Click on this button to zoom out on the view of the drawing. Zooms out by 25% of the drawing's current size each time the button is clicked.

Fit. Returns the drawing to its original view.

Undo. The undo feature is active in this screen. Clicking this will undo one step for each time it is clicked while still in the same event group (event groups are described in section 3.6.3 below). The Undo button will only work on the current event type. For example, if you are selecting a group of drill events, you can undo as many drill events while the drill event is still active. Once you leave the event, you cannot undo the selections you made under the event.

Event Data. Clicking on the Event button will change the screen to show all the prompts for the event. This is where you complete the programming of the event by adding data like RPM, Feed Rate, Tool Number and Z-axis dimensions that are not available from the CAD drawing. When you pick your event and geometry, click the Event key to fill in the rest of the data needed.

3.6.3 Event Groups

You create event groups out of items on the drawing that you want to machine the same way. Click on an event and then click all the similar pieces of geometry. When you press the Event key, the DXF Converter will automatically fill in the data in all the events in the event group by using the data you enter in the first event of an event group.

For example, say you have 5 holes to drill that are all the same size. To program these, select Drill and click on each of the five circles on the drawing that represent the drill locations. By doing this, you have created an event group made up of these five drill events. Now click on the Event button. The screen will change to show the prompts for the first event of this event group. When you fill in the blank prompts (the X and Y data is already there for you), the data you fill in will automatically be inserted into the other four events in the event group.

You can tell you are selecting items for the same event group by the color. When each item is selected, it turns purple.

If you want to separate event groups, simply select a different event between them. The other way to signal that you have finished selecting items for the event group is to click on the Event button. When you do either of these things, the items will go from purple to blue, showing that you are no longer picking items for the event group.

Continuing with the example above, say that you want to counter bore the same five locations. The locations are the same, but you want to use a different tool, different RPMs and different Z dimensions. Therefore, you will establish a different event group. After you establish the first event group you can either click the Event button and program the data for the first group, click on the drill softkey again, or click on something other than Drill, and then select Drill again. You will now be working on a new event group.

3.6.4 Chaining

Chaining is a powerful feature of the DXF Converter that enables you to program pockets and profiles quickly and easily. You will be asked if you want to chain when you select Pocket or Profile. To use chaining select Yes, then click on the first line in the pocket or profile. The first line you select will determine where the ProtoTRAK begins to machine. You will then be prompted to continue. This second line tells the ProtoTRAK which direction to go.

Once the chaining begins, it will continue in the direction you choose until it encounters one of the following:

- An end point that is the same as the beginning of the profile or pocket event. This signals the completion of an enclosed shape.

- A gap that exceeds the gap tolerance (see Section 3.3.3).

- An intersection or branch that requires a decision from you. If this is encountered, the system will prompt you to select the direction by clicking the next line or arc segment you want included in the pocket or profile.

To end the chaining process on a profile that does not close (does not have the same start and end point) simply select the Event button to fill in the additional event data, select the same event type or another event type. You can use the Undo button to un-select any number of steps in the chain, one step at a time.

3.6.5 Color Schemes and Status Flags

Through out the conversion process geometry changes colors from white, to purple, to blue to green.

White – Geometry not yet associated with any ProtoTRAK event.

Purple – Geometry has been associated with a ProtoTRAK event; the event type is still current and the Undo feature active for that event.

Blue – Geometry has been associated with a ProtoTRAK event; the event type is no longer current (you changed event types) or the Event button has been pressed. Additional information needs to be entered before the event is complete. Undo feature does not work on the event.

Green – The event has been completely defined, no further input is needed.

When you click on the Event button to enter data necessary to complete an event, you will notice the "Not OK" flag at the top right-hand corner. Once the event definition is complete, the flag changes to "OK".

This geometry color and flag status scheme will be the same for all events throughout the entire DXF converter process.

3.6.6 Completing the Event

Once you choose the event type and then click on the features on the print, there are two ways to fill in the rest of the information like feed rates and tool numbers:

1. While still in the DXF Converter, click on Event.
2. End the DXF Converter and go into Program mode.

Both ways provide you with the familiar, prompted interface of the ProtoTRAK, but we recommend that you choose the first way – stay in the DXF Converter and click the Event button.

We have two reasons for this recommendation. First, we have noticed that when we use the DXF Converter with large programs, we find that answering the prompts right away makes it a bit easier to keep things straight. Defining the event in its entirety before moving to the next event makes it unnecessary to remember event numbers later on.

The second reason is that using the DXF Converter allows you to establish event groups (event groups are described above). Event groups make it easy to program like pieces of geometry. For example, say you wanted to machine five holes the same size. Within the DXF Converter you would select Drill, then select each of the holes to add them to your event group. Then you would click on Event and fill in the prompts. The information that you put in the prompts for the first of the Drill events would be copied to the rest of the events in the event group. In contrast, if you end the DXF Converter and go to the Program Mode to fill in the prompts, you would have to fill in the same information for every drill event.

3.7 Sample Exercise

If you haven't already done it, open the file named *BALLPARK.DXF,* located in the *SWI TEST PROGRAMS* folder inside the PT4 directory. Use the Show/Hide command on the layers of this part program until it looks like Figure 3.2.2 above.

Continue and use the center of the circle as the absolute 0 reference point (see section 3.4.3).

Now you can start programming:

1. Select Drill. Note how the Drill softkey changes to simulate a pressed-in look. That means the items you click on the screen will be converted into ProtoTRAK Drill events.

2. Click the small circles the make up the bolt hole pattern one by one. Notice how the color changes from white to blue. Remember that events will be machined in the order that they are selected.

Note: whenever an event type has been selected and is active, on the lower left hand corner you will see the mouse pointers' current X and Y location in relation to the absolute zero reference point.

3. After you have clicked on all five of the circles in the bolt hole pattern, click on the Event button.

4. The prompts for the first event in the event group will appear and the NOT OK flag will appear at the top of the Event. Fill in the prompts. When the last prompt is filled in the flag changes to All OK. This means that the data you filled in has been inserted in all the rest of the Drill events that define the event group. Notice that the bolt hole pattern has turned from blue to green. This signals that these events are fully programmed.

5. Now click the Event button again. This will return you to the full screen of the drawing and prompt you to Select a new event. If you want to program the same holes again for

a different tool (like a counter bore) you would just start again at step 1 above. This starts a new event group that you will define with different information.

6. Click Pocket to program the larger circle in the middle of the part. You will be asked if you want to chain. Since the pocket is defined by a circle it doesn't matter if you click Yes or No, chaining is not relevant. Click one of them now.

7. Click on the circle. It changes to purple, indicating that it is now a Pocket event.

8. Click Event and answer the prompts to complete the event.

9. Click Event again to get back to the full screen view of the geometry and the Select Event prompt. Now all the interior features of the part should be green and the periphery white.

10. Click on Profile. It will ask you if you want to chain. Click Yes.

Note: *Chaining simply defines your event groups automatically. If you want to define them yourself, click No. The only difference is that you will have to click on all the lines and arcs in succession. For a small profile like this, it is not a big deal. A profile made up of dozens of lines and arcs would make the chaining feature more useful.*

11. Click on the first line of the profile. The ProtoTRAK will machine the part in the order you select the events.

12. Click on the second line. This will tell the DXF Converter which direction you want to go. Once you do that, the rest of the profile is done for you.

13. Click Event and fill in the prompts.

14. Click Event again. You will be back at the full screen graphics and the entire part will be drawn in green.

15. Click End DXF. The DXF Converter turns off and the system reverts back to regular ProtoTRAK operation.

16. You may want to save the part program under a different name after it has been converted. In the Program In/Out Mode, press Save. The file will be the same name, unless you change it, but the extension (shown in SAVE AS) will now be our in our ProtoTRAK format, which saves the events you just created. You can also change the file name in the Program Mode.

3.8 Creating a DXF File from a ProtoTRAK Program

When you have a program in the current memory of your ProtoTRAK control, you can save the program geometry as a DXF file. The DXF file can then be transferred to a CAD system to note any shop floor changes made, or as a reverse engineering tool.

4.0 Using the DXF Converter for Lathes

Using the DXF Converter is a matter of a few simple steps that will be described in this section.

Please set the ProtoTRAK CNC or PT4OL to the desired Inch or Metric mode before entering the DXF Converter. These settings cannot be changed once the converter process has started, until exiting back to the main ProtoTRAK screen.

Your ProtoTRAK software comes with a sample DXF program you can use to follow along with these instructions. This program is named DXFLatheSample1.DXF and it is located on the C drive in the ProtoTRAK CNC in the directory: C:\PT4\DXF Samples\Lathe Samples. For the Offline program, the program is in the drive you specified when you installed the ProtoTRAK Offline on your computer.

4.1 Opening a DXF Program

The DXF Converter is launched automatically when you open a file with the DXF or DWG extension.

When you select a file with an extension of .DXF or .DWG, the Open As field automatically changes to ACAD DXF. Press Open File and the DXF Converter will begin to process the file.

Note: *You can use Mode (Alt + M in Offline) to exit DXF and return to normal ProtoTRAK operation at any time during the DXF conversion process. This will discard any information generated during the DXF conversion session.*

4.2 Functions of the Status Line

During the conversion process, there are some new buttons that don't appear elsewhere on the ProtoTRAK.

Items unique to the DXF Converter appear in the status line at the top:

Background Selection Button. Click on this to toggle between a white and black background. You should use the background that allows you to see the layers on the drawing. So that they remain visible, the DXF Converter will automatically change the color of white lines to black or black lines to white depending on the background you choose. Colors used in the drawing that are other than black or white will stay the same regardless of background.

Zoom In, Zoom Out and Auto-Fit Drawing Buttons.

Zoom In. Click on this button to zoom in on the view of the drawing. Zooms in by 25% of the drawing's current size each time the button is clicked.

Zoom Out. Click on this button to zoom out on the view of the drawing. Zooms out by 25% of the drawing's current size each time the button is clicked.

Fit. Returns the drawing to its original view.

Undo. The undo feature is active at logical points in the conversion process. Clicking this will undo one step for each time it is clicked when making simple edits to your drawing or defining events (see sections 4.7 & 4.8). The Undo button will only work on the current event type. For example, if you are selecting a group of turn events, you can undo as many turn events while the turn event
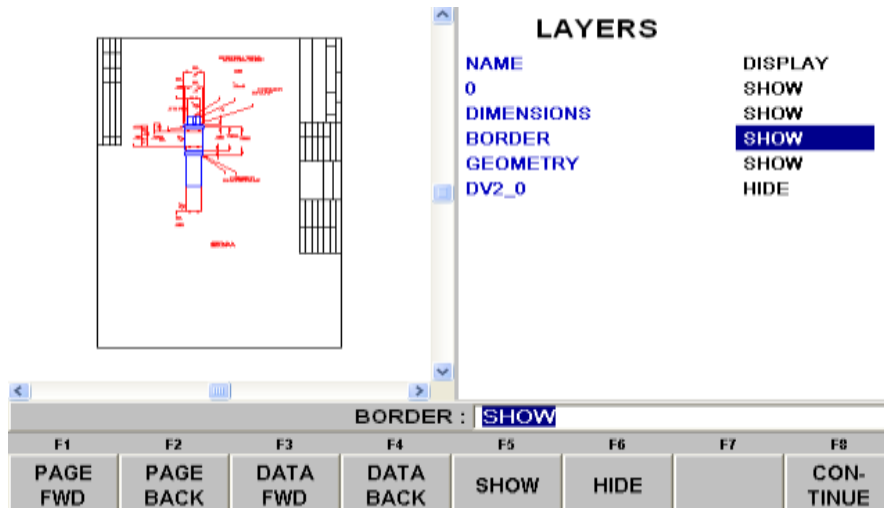
is still active. Once you leave the event, you cannot undo the selections you made under the event.

Event Data. This button only appears in the last screen in the conversion process (section 4.8). Clicking on the Event button will change the screen to show all the prompts for the event. This is where you complete the programming of the event by adding data like RPM, Feed Rate, Tool Number and other non-dimensional data not available in the drawing. When you pick your event and geometry, click the Event key to fill in the rest of the data needed.

4.3   Choose the Layers That You Want Displayed

CAD files are made for printing, not machining. Therefore, DXF and DWG files usually contain information that is good to have on a print, but not useful for machining. Examples include borders, writing and dimension lines. To make the DXF conversion convenient, it is helpful to eliminate those unneeded elements of the drawing from view.

The initial screen of the DXF converter has all the drawing's layers visible. This step allows you to Hide unwanted information so you may focus on what you want to machine. See figures below.

FIGURE 4.3.1 The first screen you see when the DXF Converter launches. Notice that the entire drawing, borders and dimensions, is shown on this first view

FIGURE 4.3.2 Through a process of elimination, we have isolated the view of what we want to machine.

To choose a layer that you would like to Hide use the Data Forward key or simply click on the layer name with the mouse. You can go between Show and Hide as often as you like as you choose which layers to eliminate from view.

Press Continue when only the layer(s) you wish to remain are shown. When you leave this screen, you may return to it by pressing the Back key (Alt + B on the PT4OL). However, any additional data you have added will be lost.

4.4   The Gap Tolerance

The ProtoTRAK makes it easy to clear the stock with the powerful Cycle event feature and the DXF Converter makes it even easier by providing the dimensions. One potential problem in this process is that a CAD drawing is made to be looked at, not machined. DXF and DWG files may contain lines or arcs that look connected on a print, but really have tiny gaps between them. The gap tolerance of the DXF Converter is an elegant solution to this problem. You enter a single value – a tolerance – and the DXF Converter automatically closes the gaps that are within that value while you are programming. If you don't want this done automatically, you aren't forced to apply the tolerance. The DXF Converter will prompt you when it encounters a gap and let you decide whether to close it.

4.4.1 Gap Tolerance Applied Automatically

As soon as you decide your layers and select Continue, you will be asked about the Gap tolerance. If you want to use the 0.005" or 0.15mm gap tolerance in the status line, simply choose YES. You may also input a different value for the gap tolerance and then choose YES. The DXF Converter will refer to this tolerance and will automatically stretch lines an amount that is within the tolerance in order to make them connect. If a gap is encountered that exceeds this tolerance, you will be prompted with a choice to close the gap that is encountered.

If you choose NO, the DXF Converter will prompt you every time the gap is encountered while programming a cycle.

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4.4.2 Closing Gaps As They Are Identified During Programming

Once you begin the programming of a cycle, you will be prompted to resolve a gap between lines and arcs that are to be connected when:

- You input a gap tolerance but the gap that is encountered exceeds the tolerance you entered.

- You chose NO to the gap tolerance.

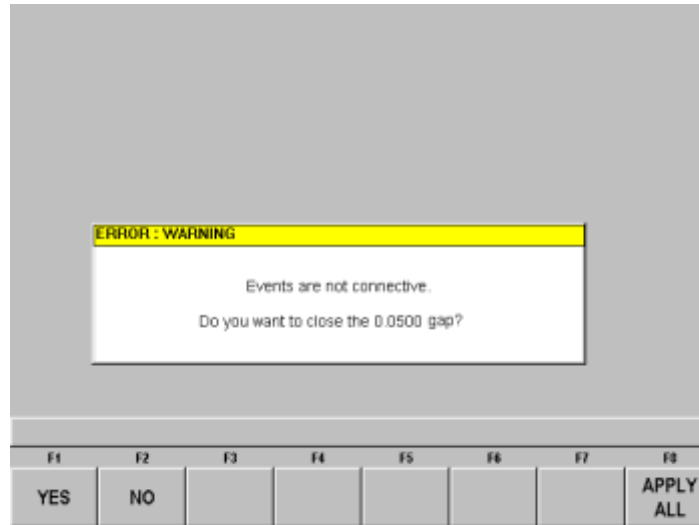In either case, when a gap is encountered, the DXF Converter will display a message like the following:

FIGURE 4.4.2 A gap is encountered that exceeds the gap tolerance.

- If YES is chosen, the displayed gap value will only be used to close the current gap. The original gap tolerance value (if any) will be retained for checking all other gaps encountered.

- If NO is chosen, the gap will not be closed. The original gap tolerance value will be retained and drawing segment will remain unselected.

- If Apply All is chosen, the gap will be closed and this new gap value will replace the previous one as the new gap tolerance value. You will not be prompted again unless a gap outside this new tolerance is encountered.

4.4.3 Gap Tolerance When Chaining

When the chaining capability is used (chaining is described in Section 4.8.2 below) the gap tolerance is applied. When a gap that exceeds the gap tolerance is encountered in the chaining process, the DXF Converter will assume it has reached the end of the geometry to be chained. In order to have the chaining continue, simply click on the next geometry to be connected. You will then get the error message shown in figure 4.4.2 with the choice of changing the gap tolerance with the Apply All command.

4.5   Rotate Drawing and Pick a Zero Absolute Reference

4.5.1 Rotate 90 Degrees

Sometimes, the drawing you convert will not be in the correct orientation for proper machining on a lathe. Use the "Rotate 90 degrees" button to rotate your drawing 90° in the counter clock-wise direction each time it is pressed.
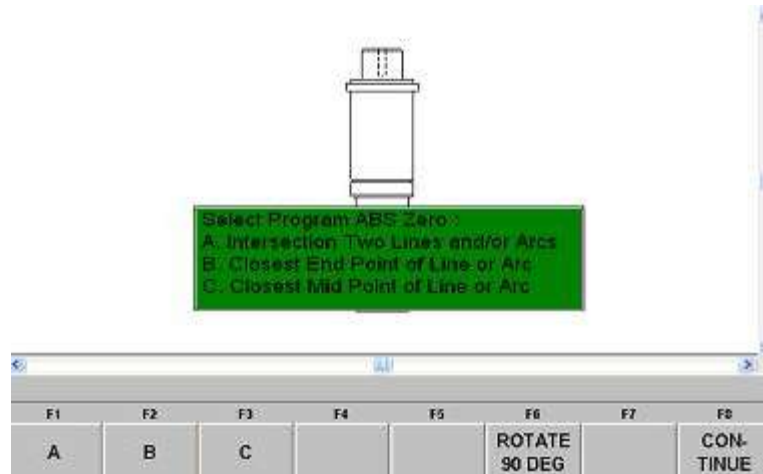
FIGURE 4.5.1 We will have to press the Rotate button a total of 3 times to get the drawing in the correct orientation.
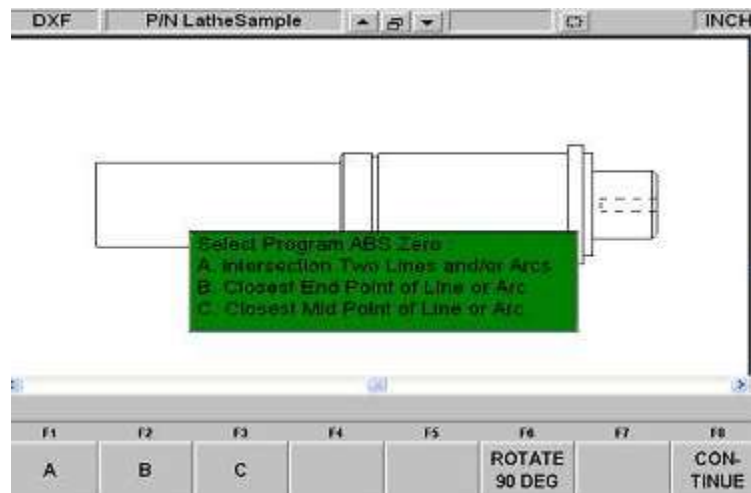

FIGURE 4.5.2 The drawing after it has been rotated to the correct orientation.

The drawing file you converted has its own reference point, but that may not be the most convenient one for programming. The screen you encounter after setting the gap tolerance will allow you to select your own Absolute zero reference. The DXF Converter provides three choices for choosing the Absolute zero reference point that are described below.

4.5.2 Choice A: Intersection of Two Lines or Arcs

This allows you to use intersecting points, like corners, as the absolute zero reference. The intersecting points can be taken from two lines, two arcs, or a combination of a line and an arc. Circles cannot be used with this choice.

To make an intersection the absolute reference, select choice A, then click on the two pieces of geometry that will intersect at the point you want to use. After you click on the second piece of geometry, a red x will appear to show you where the absolute zero is now set.

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This choice also allows the use of the intersection of two lines that meet off of the given geometry. If this is the case, you will not be able to see the red x where the new absolute zero reference is set.

4.5.3 Choice B: Closest Endpoint of Line or Arc

This allows you to the use an end point of a line or an arc. Click on a line or arc close to the end point to be used as the absolute zero reference. Circles cannot be used with this choice.

4.5.4 Choice C: Closest Midpoint of Line or Arc

This allows you to use the midpoint of a line or arc as the absolute reference point by simply clicking on a line or arc.

FIGURE 4.5.4 For this exercise, our best option is to go with C, Midpoint of line or arc and click on the line at the very front of the drawing.

4.5.5 Continue: Use the Drawing's Absolute Zero

If CONTINUE is pressed without selecting an absolute zero reference point with one of the methods above, then the reference point given in the drawing will be used. This can be helpful if the CAD programmer used a more appropriate absolute zero reference than one that can be created with the given choices. However, in most cases, when CAD drawings are created, the absolute zero reference is usually in one of the corners of the drawing which does not make the best ABS 0 reference point for machining purposes.

This last option is not available if the drawing has been rotated (like in the example above). If you rotate the drawing and press continue without selecting an alternate ABS 0 reference point, you will receive the following error message. Press Resume, you will be return to the ABS 0 selection screen where you can select a reference point.
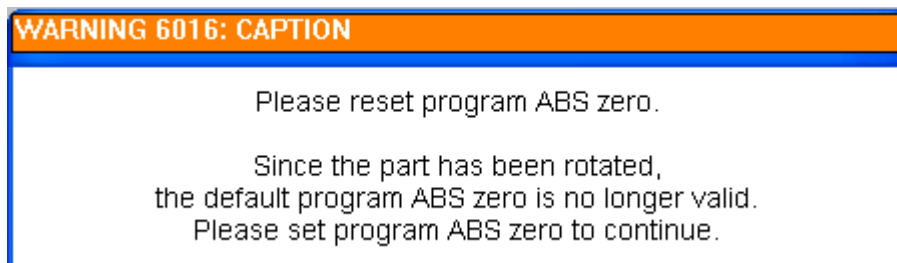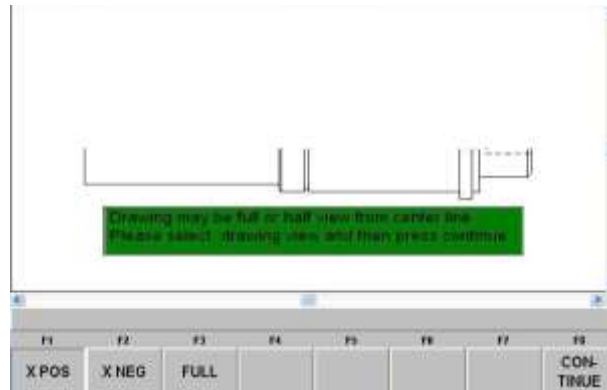

Figure 4.5.5 Error message received when the drawing has been rotated and no alternate ABS 0 reference has been selected.

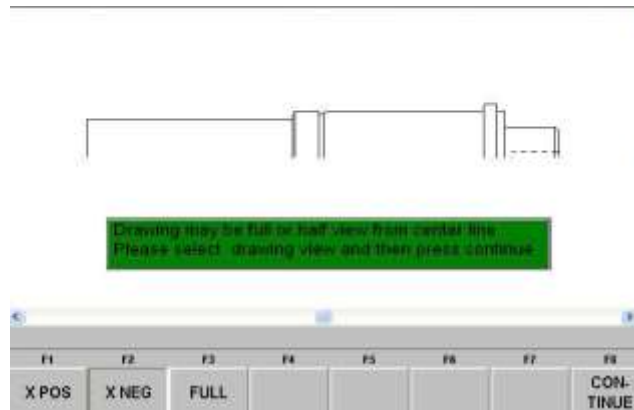*We recommend that you use the centerline of the part as the ABS 0 reference point.*

Note: *The back key (Alt + B) can be used to return to this screen from any subsequent screens, however, programmed information will be lost.*

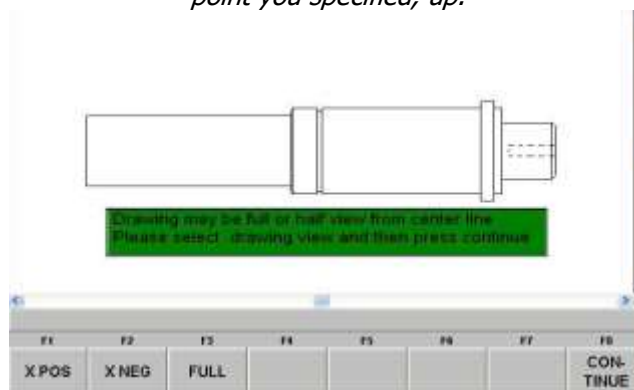## 4.6   Positive Half, Negative Half and Full Drawing Display

The ProtoTRAK DXF Lathe converter allows you to display the Positive X potion of the drawing, the negative X portion of the drawing or the entire drawing. The converter will use the ABS 0 you specified to determine where to divide the positive or negative side of the drawing.

FIGURE 4.6.1 The X Pos button will display the portion of the drawing from the ABS 0 point you specified, down.

FIGURE 4.6.2 The X neg button will display the portion of the drawing from the ABS 0 point you specified, up.

FIGURE 4.6.3 The FULL button, will restore the drawing to its entirety.

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4.7 Edit Drawing

There will be times where the geometry on your drawing is not laid out in the best way possible to create a tool path. In some instances, you might be missing a line in order to properly create a tool path. In other cases, you might need to get rid of a line in order to create a proper tool path. The ProtoTRAK Lathe DXF converter allows you to make simple edits to your drawing in order to accommodate those situations.

Note: These edits will not be saved onto your original drawing file.

Figure 4.7 The Edit Screen

4.7.1 Add Line

The Add Line feature allows you to add a new line to your drawing by clicking two existing end points.

To add a line using the Add Line feature, first select Add Line, then click on the ending point from which you'd like to start the line.

Note: The Add Line feature will select the endpoint closest to where you click on the line.
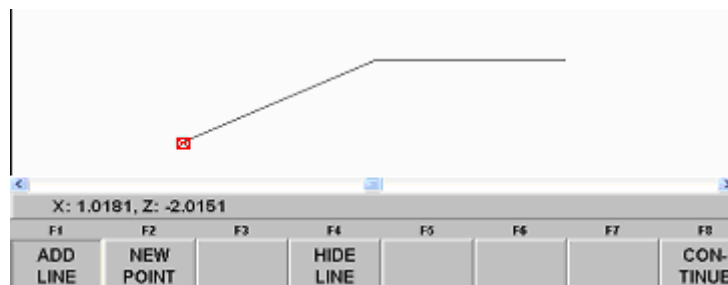

Figure 4.7.1. First click a line where you want to start

Then select the second end point at which you'd like the line to end.

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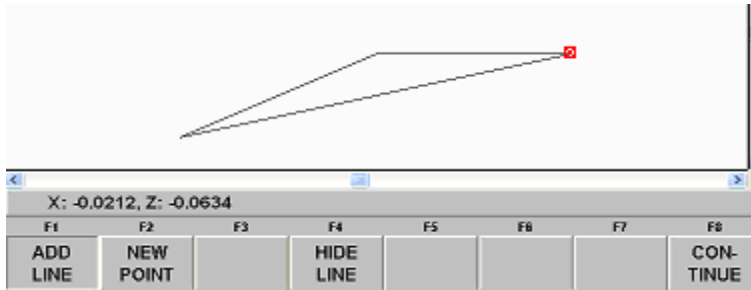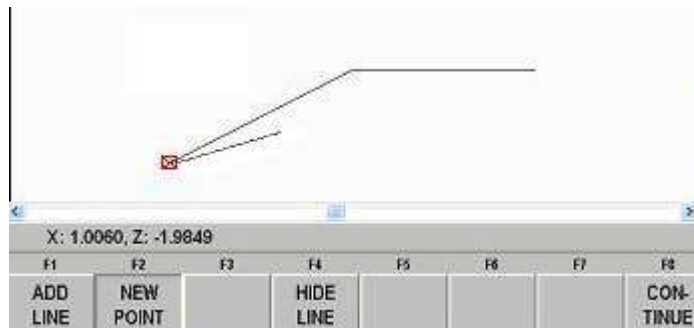The ProtoTRAK DXF Converter Operating Manual

Figure 4.7.1.2 To complete the line, click the end point on the existing line where you want the line to end

4.7.2 New Point

The New Point feature allows you to add a new line to your existing drawing by clicking on one end point, then dragging the line with the mouse and placing the second end point where ever you like.

To add a line using the New Point feature first click on New Point, then click on the existing line where you would like the new line to begin. You will then notice that a new line is created and it will follow the movement of the mouse.

Note: *The New Point feature will select the endpoint closest to where you click on the line.*

4.7.2.1 The line starts from the end point of the existing line.

Once you have moved the mouse to the desired location click to end the line.

4.7.2.2 New Point allows you to end the new line anywhere on the drawing.

4.7.3 Hide Line

The Hide Line feature allows you to hide all or a portion of a line. This is helpful when using the Chain function (explained in section 4.8.2) on a cycle event.

To use the Hide Line feature, select Hide Line, then click on the line you would like to hide, this will highlight the entire line purple. Now, if the line is intersected at any point, click on the side or portion of the line you would like to hide and that section of the line disappears. If the line is not intersected at any point, clicking on the line a second time will hide the entire line.

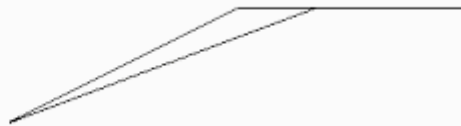*Figure 4.7.3.1   First highlight the entire line, then click on portion of line to hide.*

Figure 4.7.3.2   Top portion of the line has been hidden.

4.7.4 Using the Drawing Edit Features

The Add Line, New Point and Hide Line features help us create a proper tool path.
In our sample DXF part the majority of the part could be programmed with the Cycle event. However, there is a small groove in the middle of the drawing that would be best done with a groove tool (rather than a Right Hand Facing tool) using the Groove Event.
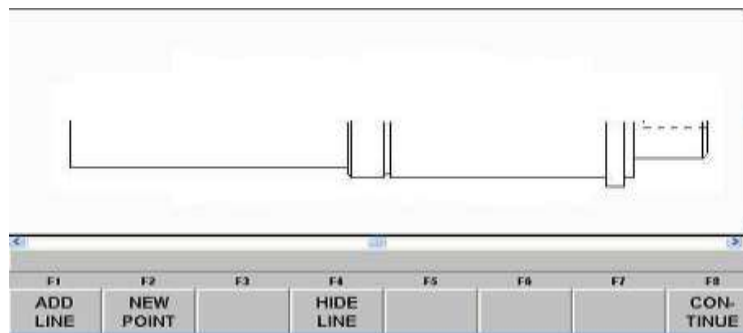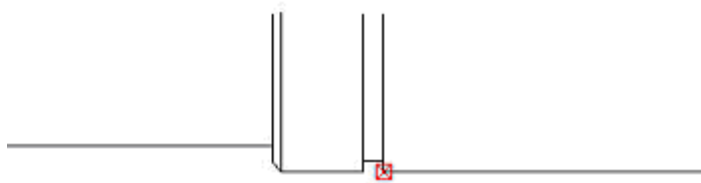

Figure 4.7.4 Positive half of the original drawing showing the groove in the middle of the part.

The strategy is to remove the rest of the material with the Cycle Event and preserve the Groove for machining with the Groove Event later. To do that select Add Line and use the zoom feature to get a good view of the lines. Then, click on the horizontal line to the right of the groove.
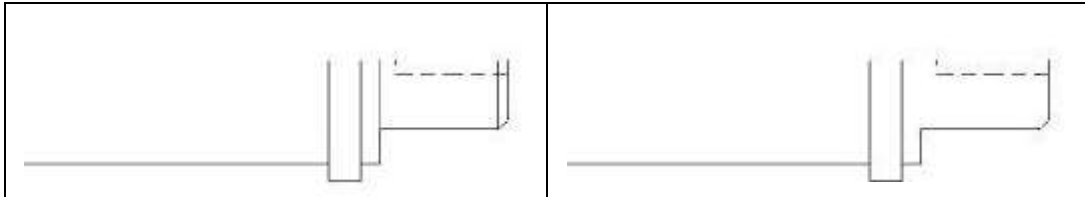
Figure 4.7.4.1 Use the Add Line feature to save the material for machining in a separate operation.

When you click on the horizontal line to the left of the groove the new line is added.

Figure 4.7.4.2 The new line is inserted.

The vertical lines at each radius make it difficult to create a tool path, and are not needed to create a part profile. Use the Hide Line feature to get rid of the lines.

Figure 4.7.4.3   Use the Hide Line feature to eliminate drawing lines that are not needed.

4.8   Converting Geometry into a ProtoTRAK Program

After you have edited and simplified the geometry, programming is just a matter of selecting the events. With the ProtoTRAK DXF Converter, you have total control over how the part is made - just like regular ProtoTRAK programming.

The basic steps are:
1.   Pick the event type.

2.   Pick the geometry on the drawing. This will load the dimensions from the drawing into the event type you choose.

3.   Fill in the rest of the prompts to complete the event.

4.8.1 Event Types

Posn. Allows the creation of a position event by clicking anywhere on the screen. You will see a small circle, or dot, appear in the location where you clicked.

Drill. The Drill event converts any horizontal line into a drill event. The converter will take the most positive Z-dimension as the entry of the drill (it will also add .100" as a safety rapid point) and will use the most negative Z-dimension as the depth of the drill. X will always be at zero.

Bore. The Bore event converts any horizontal line into a Bore event. The converter will take the most positive Z-dimension as the entry of the drill (it will also add .100" as a safety rapid point) and will use the most negative Z-dimension as the depth of the Bore.

Turn. Converts any straight line into a Turn event. Click on the end of the line where you want to establish the beginning point (X Beg, Y Beg).
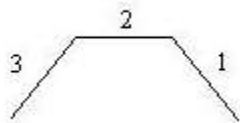
Arc. Converts any arc in the drawing into an Arc event. Click on the end on the arc where you want to establish the beginning point (X Beg, Y Beg).

Cycle. Converts connecting lines and arcs into a Cycle event. When selecting a Cycle event, the first line clicked determines where the Cycle starts and the second line clicked determines the direction of the Cycle.
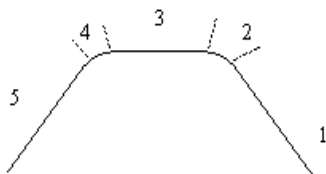
X stock. The cycle event inside of the DXF converter (only) has an X stock field. This field allows you to enter the radius of the stock you will be using and helps the ProtoTRAK determine how much material needs to be roughed out.

Thread. The information provided in drawings is insufficient to properly create a ProtoTRAK thread event. The thread event inside the DXF converter will not allow you to select any geometry to transfer coordinates. Instead, select Thread to insert the Thread event into the program and manually fill in the data to complete the event.
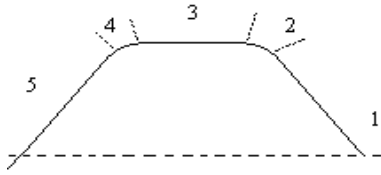
Groove. Allows the conversion of three lines, in a specific order, into a groove event.

In this drawing, for example, we would first click on Groove, and then we would click on the three lines in a counter clock-wise order. The order in which these lines are selected will determine the direction in which the event is roughed and finished.

The event only allows for the selection of three lines. When we want to include conrads and chamfers, in the example to the left, we would select Groove then click on lines 1, 3 and 5 skipping lines/arcs 2 and 4. The conrads or chamfers can then be entered manually into the event.

The groove must also begin and end at the same X dimension. In the sample to the left, the converter would give an error when selecting these three lines as a groove event because line 5 does not end at the same x dimension as line 1 does. The Add Line feature could be used in this case.

4.8.2 Chaining

Chaining is a powerful feature of the DXF Converter that enables you to program Cycles quickly and easily. You will be asked if you want to chain when you select Cycle. To use chaining select Yes, then click on the first line of the Cycle. The first line you select will determine the starting point of the cycle. You will then be prompted to continue. This second line tells the ProtoTRAK which direction to go.

Once the chaining begins, it will continue in the direction you choose until it encounters one of the following:

- A gap that exceeds the gap tolerance (see Section 4.4.3).
- An intersection or branch that requires a decision from you. If this is encountered, the system will prompt you to select the direction by clicking the next line or arc segment you want included in the pocket or profile.

To end the chaining process on a Cycle, simply select the Event button to fill in the additional event data or select the same or a different event type. You can use the Undo button to un-select any number of steps in the chain, one step at a time.

4.8.3 Color Schemes and Status Flags

Through out the conversion process geometry changes colors from white, to purple, to blue to green.

White – Geometry not yet associated with any ProtoTRAK event.

Purple – Geometry has been associated with a ProtoTRAK event; the event type is still current and the Undo feature active for that event.

Blue – Geometry has been associated with a ProtoTRAK event; the event type is no longer current (you changed event types) or the Event button has been pressed. Additional information needs to be entered before the event is complete. Undo feature does not work on the event.

Green – The event has been completely defined; no further input is needed.

When you click on the Event button to enter data necessary to complete an event, you will notice the "Not OK" flag at the top right-hand corner. Once the event definition is complete, the flag changes to "OK".

This geometry color and flag status scheme will be the same for all events throughout the entire DXF converter process.

4.8.4 Completing the Event

Once you choose the event type and then click on the features on the print, there are two ways to fill in the rest of the information such as feed rates and tool numbers:

- While still in the DXF Converter, click on Event.
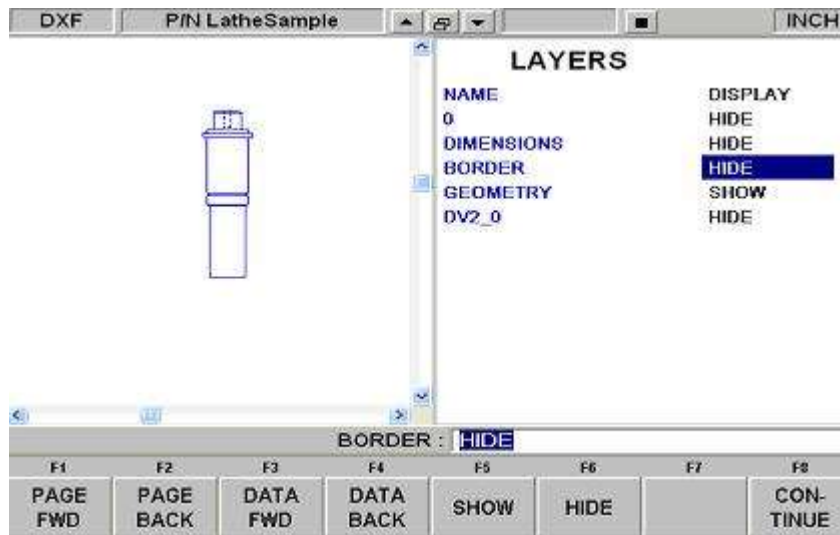- End the DXF Converter and go into Program mode.

Both ways provide you with the familiar, prompted interface of the ProtoTRAK, but we recommend that you choose the first way – stay in the DXF Converter and click the Event button.

We have noticed that when we use the DXF Converter with large programs, we find that answering the prompts right away makes it a bit easier to keep things straight. Defining the event in its entirety before moving to the next event makes it unnecessary to remember event numbers later on.
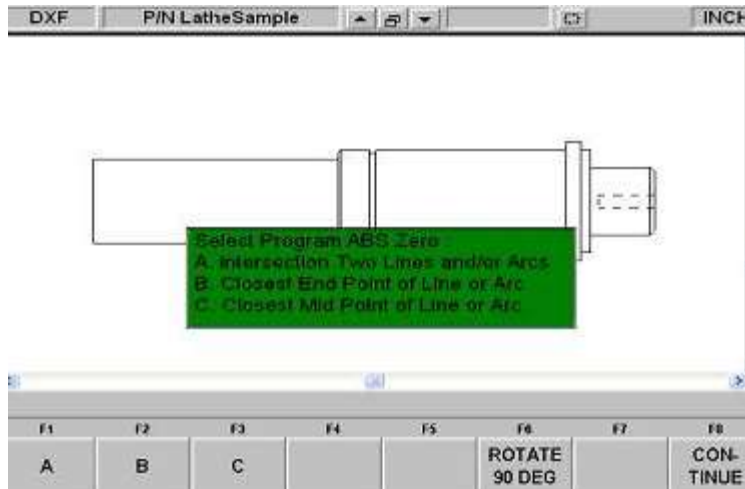
4.9 Sample Exercise

Open the program named DXFLatheSample1.dxf that is located on the C drive in the ProtoTRAK CNC in the directory: C:\PT4\DXF Samples\Lathe Samples. For the Offline program, the program is in the drive you specified when you installed the ProtoTRAK Offline on your computer.

Hide all the layers until only the needed geometry is visible, press continue.
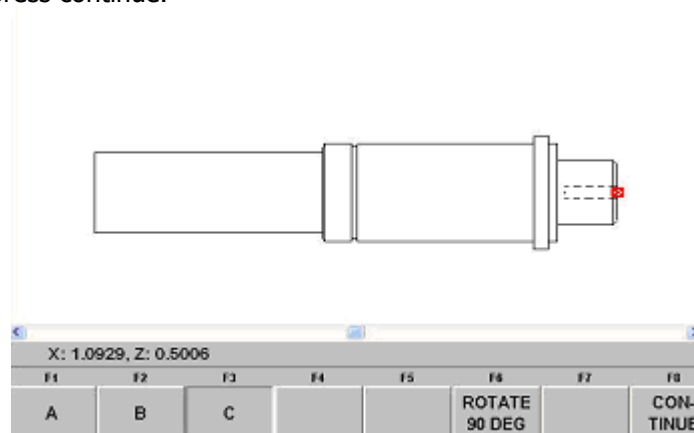

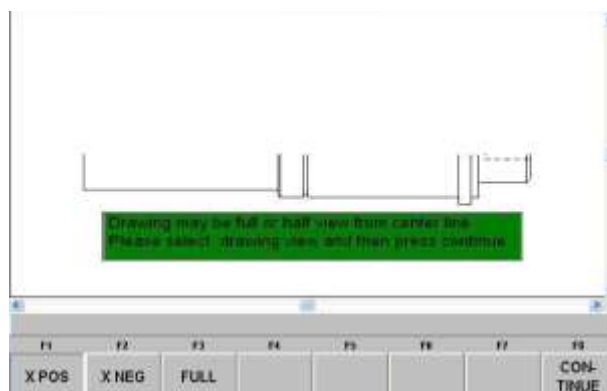Press Yes at the Gap Tolerance screen to enter the default gap tolerance value.

Press the Rotate 90 Deg button three times to get the drawing in the proper orientation.

Select option C and click on the vertical line on the drawing that is furthest to the right as your ABS 0 reference, press continue.
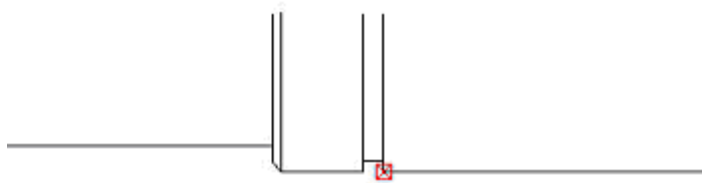

Press the X POS button to display the positive half of the drawing. Press Continue.

Use the Zoom feature to get a better view of the groove in the middle of the drawing. Select Add Line, click on the horizontal line to the right (it really does not matter which line, left or right, you select first).
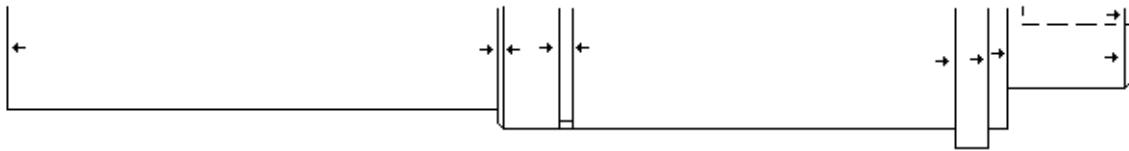
Click on the horizontal line to the left, you will see an new line appear.

Press Hide Line. Look at the drawing below, click twice on each line that has an arrow next to it, in the general area of the arrow. One click is to select the line; the second click is to hide that portion of the line.
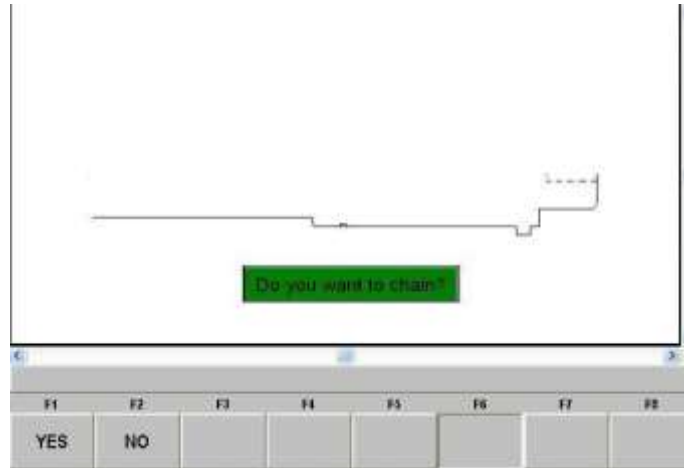

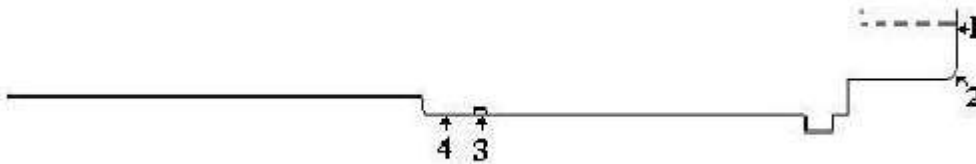After you have hidden all the lines, the part should look like the one below. Press Continue.

We are now in the tool path definition stage of the DXF conversion. Select Cycle.

As soon as we select the Cycle event, we receive a message, which reads, "Do you want to chain?" Select Yes.
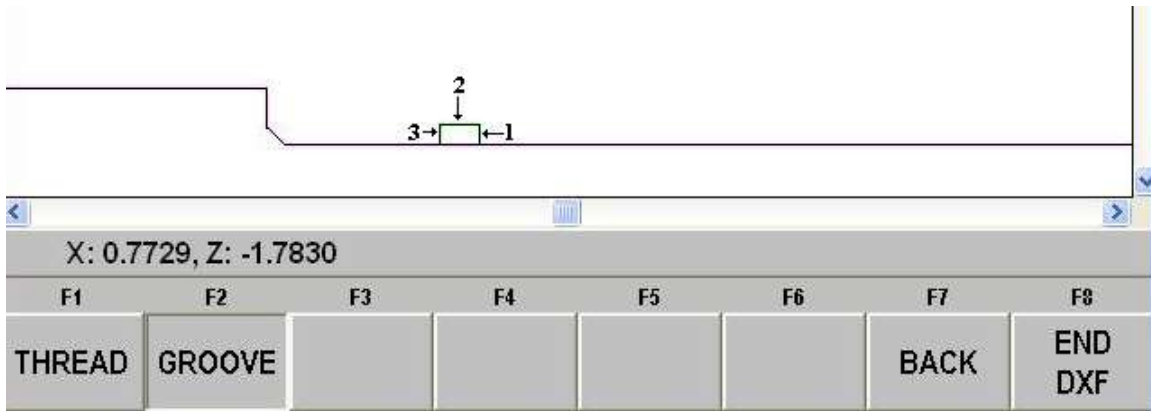
Note: *Chaining simply defines your event groups automatically. If you want to define them yourself, click No. The only difference is that you will have to click on all the lines and arcs in succession. The more features a part has, the more time the Chaining feature will save you.*

1. After selecting yes, click on the line at the front of the part that we chose as the ABS 0 reference You will receive a message asking you "Please click to continue". Click on the slanted line.
2. That follows the first line. You will receive another "Please click to continue" message, click on the line we created earlier.
3. You will immediately receive another "Click to continue" message, click on the line following it.
4. And the chain will continue to the end of the part.

Now we click on the EVENT button on the upper right hand corner of the screen. Fill in all the necessary prompts (RPM, federate, tool #, etc.), when done click on the Event button a second time to return to the event definition screen.

Now we want to define the Groove event for the small groove on the part. First click on More, then Groove. We will now use the zoom function and zoom in on the groove to have a better view of the lines we want to pick. We will first click on the vertical line 1, then 2 and finally 3.

X: 0.7729, Z: -1.7830

F1	F2	F3	F4	F5	F6	F7	F8
THREAD	GROOVE					BACK	END DXF

Once you have clicked all three lines, click on the Event box and fill in the required data. When done click the Event button a second time to return to the event definition screen.

Use the zoom function to get a better view of the front of the part. Select Drill and click anywhere on the vertical line at the front of the part.

Click on Event and you will notice that the drill event only uses the Z-axis information and does not care about the location of the X-axis. Also, although the beginning of the drill should be at Z 0.0, the converter has automatically added a .100" safety/rapid point.

Fill in all the required data for the drill event and Click Event. All the geometry should now be a color green, which means we are ready to return back to regular ProtoTRAK operation.

Click Event again to get back to the full screen view of the geometry and the Select Event prompt. Now all the interior features of the part should be green and the periphery white.

You may want to save the program under a different name after it has been converted. In the Program In/Out Mode, press Save. The file will be the same name, unless you change it, but the extension (shown in SAVE AS) will now be PT4. You can also change the file name in the Program Mode.

4.10 Creating a DXF File from a ProtoTRAK Program

When you have a program in the current memory of your ProtoTRAK control, you can save the program geometry as a DXF file. The DXF file can then be transferred to a CAD system to note any shop floor changes made, or as a reverse engineering tool.

5.0 Tips

We are using the data contained in a CAD file to make programming easier. However, we must keep in mind that the original purpose of the CAD file is to make a print, not a program.

This section describes some of the things that can be done in making the CAD file that will make the conversion go better.

5.1   Tips for Milling DXF

Limit to One Z Plane
Geometry on one layer cannot be connected to geometry on a different layer. When drawings contain geometry in multiple Z-axis layers, a 2D view can make it look like the geometries are connected when they are not because they are in different layers. You may not be aware of it until you try to connect two lines or arcs and receive an error message.

To Resolve Problems Hiding Layers
There are intermittent instances where files saved as .DWG will have layering problems when brought into the DXF converter. If your file experiences layering problems, simply have your CAD programmer open it with AutoCAD and save it as .DXF or an earlier version of AutoCAD (i.e. AutoCAD 2000 to Version 14).

Reducing File Loading Wait Time
The amount of time you have to wait for the CAD drawing to load is a function of the number of entities found in your drawing. To lessen wait time, try eliminating un-needed geometry from your drawing. This will be most beneficial when using the Chaining feature.

Layer to Isolate Geometry
Organize drawing layers so that information that is not needed for the conversion process can be hidden. You want to be able to isolate the geometry to be programmed from the other things like page borders that appear in a drawing file.

Avoid Complicated Graphics in Your Title Block
Graphics like company logos may involve a large number of geometric entities. If this is not on a layer that is hidden, the converter will process this as geometry that you wish to machine. This may result in a longer waiting time while your drawing is being processed. (In our beta test, we encountered one graphic that caused the DXF Converter ten minutes of processing time on one of our slower computers!).
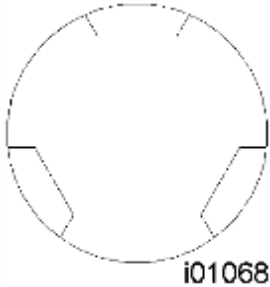
Avoid Non-Supported Drawing Objects
Only lines, arcs and circles can be converted into ProtoTRAK events. Using more complicated geometries like splines, or polylines, will cause the converter to break these entities down into lines and arcs that it can recognize. This may result in a longer waiting time while your drawing is being processed.

Avoid Large Gaps
Try to eliminate large gaps in your drawings. It is possible to slightly shift geometry when gaps are closed through the PT4 Converter process.

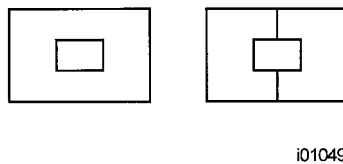**Add End Points for Connective Geometry**
The PT4 DXF Converter selects geometry from end point to end point. If you have intersecting geometry but do not add an end point at the right place, the geometry cannot be connected. In the example below in figure 5.8, the interior of the circle and the interior tabs are to be machined. If the circle is drawn as a circle and then the tabs added, the DXF Converter will not have end points to use to connect the geometry of the circle with the lines on the tabs. To program this interior, break the circle into arc segments.

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FIGURE 5.8 Break the circle into arcs to machine the interior as a Pocket or Profile.

Islands
There is not an islands selection in this version of the DXF Converter. A simple trick enables you to get the same result easily with the convenience of the Pocket programming of the DXF Converter: divide the island into two pockets by adding a line. See Figure 5.9 below.

i01049

FIGURE 5.9 Two simple lines divide the island into two pockets.

5.2   Tips for Lathe DXF

Limit Connective Geometry to One Plane
Geometry on one layer cannot be connected to geometry on a different layer. When drawings contain geometry in multiple layers, a 2D view can make it look like the geometries are connected when they are not because they are in different layers. You may not be aware of it until you try to connect two lines or arcs and receive an error message.

To Resolve Problems Hiding Layers
There are intermittent instances where files saved as .DWG will have layering problems when brought into the DXF converter. If your file experiences layering problems, simply have your CAD programmer open it with AutoCAD and save it as .DXF or an earlier version of AutoCAD (i.e. AutoCAD 2000 to Version 14).

Reducing File Loading Wait Time
The amount of time you have to wait for the CAD drawing to load is a function of the number of entities found in your drawing. To lessen wait time, try eliminating un-needed geometry from your drawing. This will be most beneficial when using the Chaining feature.

Layer to Isolate Geometry
Organize drawing layers so that information that is not needed for the conversion process can be hidden. You want to be able to isolate the geometry to be programmed from the other things like page borders that appear in a drawing file.

Avoid Complicated Graphics in Your Title Block
Graphics like company logos may involve a large number of geometric entities. If this is not on a layer that is hidden, the converter will process this as geometry that you wish to machine. This may result in a longer waiting time while your drawing is being processed. (In our beta test, we encountered one graphic that caused the DXF Converter ten minutes of processing time on one of our slower computers!).

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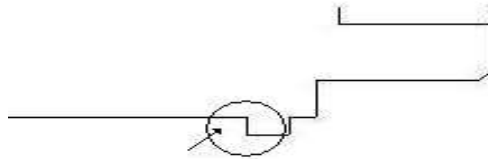
Avoid Non-Supported Drawing Objects

Only lines, arcs and circles can be converted into ProtoTRAK events. Using more complicated geometries like splines, or polylines, will cause the converter to break these entities down into lines and arcs that it can recognize. This may result in a longer waiting time while your drawing is being processed.
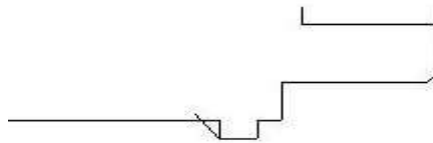
Avoid Large Gaps

Try to eliminate large gaps in your drawings. It is possible to slightly shift geometry when gaps are closed through the PT4 Converter process.

Edit Geometry to Accommodate Your Tool

In the drawing below, it would be desirable to machine most of the material in one operation using one tool with the powerful Cycle routine. A right-hand turn/face tool would be the best for most of the geometry, but clearly won't work for the last sharp corner to the left.

In this case, use the New Point feature and create a new line for a profile. The objective is to create a geometry that your right-hand turn/face tool can handle. After roughing the material, you can machine that last corner in a separate operation with a different tool.
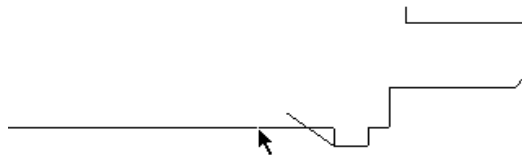
Creating New Intersection Points

The valuable chaining feature of the cycle event uses points of intersection to tell where one line ends and a different line begins. When you add a line using the New Point feature as in Section 5.8 above, the drawing file does not yet contain an intersection at the point the lines cross. The reason for this has to do with the way drawing files are created. It could be a show-stopper for using the cycle event in these cases, except that we can easily establish the intersection with a simple trick using the Hide Line feature (you may want to review how this feature works in section 4.7.3).

In the example above, we added a line that made sense for the right-hand turn/face tool we are planning to use to do our roughing.
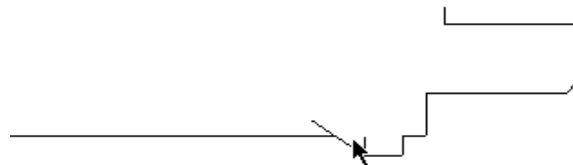
At this point, the problem is that the chaining feature will not recognize the intersection point where our new line crosses the original line. We can make it recognize it by hiding part of the original line, and then undoing this so that our original line will be preserved for machining later.

To see how this works, consider the example below. First, we select the Hide Line button. Then, we select the original line that crosses the line that we added.

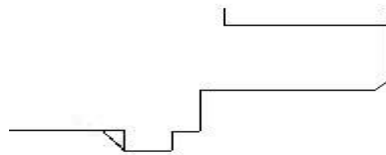*Figure 5.9.1 Select Hide Line and click on the original line.*

Next, we click on the short side of the line and it is hidden.

Figure 5.9.2 Click on the short side of the line and it is hidden.

Now, click on the Undo button to restore the short side of the line because we will need it later to program that corner with a different tool. Now the point of intersection has been established and we can use the cycle function.

Note: *Use the Hide Line function to trim the line we added before programming the cycle events.*

Figure 5.9.3 The new line segment has been properly created.

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TRAK Warranty Policy

Warranty

TRAK products are warranted to the original purchaser to be free from defects in workmanship and materials for the following periods:

Product	Warranty Period	
	Materials	**Factory Labor**
New TRAK/ProtoTRAK	1 Year	1 Year
Any EXCHANGE Unit	90 Days	90 Days

The warranty period starts on the date of the invoice to the original purchaser from Southwestern Industries, Inc. (SWI) or their authorized distributor.

If a product, subsystem or component proves to be defective in workmanship and fails within the warranty period, it will be repaired or exchanged at our option for a properly functioning unit in similar or better condition. Such repairs or exchanges will be made FOB Factory/Los Angeles or the location of our nearest factory representative or authorized distributor.

Warranty Disclaimers

- This warranty is expressly in lieu of any other warranties, express or implied, including any implied warranty of merchantability or fitness for a particular purpose, and of any other obligations or liability on the part of SWI (or any producing entity, if different).

- Warranty repairs/exchanges do not cover incidental costs such as installation, labor, freight, etc.

- SWI is not responsible for consequential damages from use or misuse of any of its products.

- TRAK products are precision mechanical/electromechanical/electronic systems and must be given the reasonable care that these types of products require. Evidence that the product does not receive adequate Preventative Maintenance may invalidate the warranty. Excessive chips built up around ballscrews and way surfaces is an example of this evidence.

- Accidental damage, beyond the control of SWI, is not covered by the warranty. Thus, the warranty does not apply if a product has been abused, dropped, hit or disassembled.

- Improper installation by or at the direction of the customer in such a way that the product consequently fails, is considered to be beyond the control of the manufacturer and outside the scope of the warranty.

- Warranty does not cover wear items that are consumed under normal use of the product. These items include, but are not limited to: windows, bellows, wipers, filters, drawbars and belts.

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