

The ProtoTRAK®RX DXF Converter Operating Manual

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**Converter part numbers: CONV-DXF-RMX
CONV-DXF-RLX**

DXF Converter software for Mills and Lathes
Compatible with PTRXOL offline and
ProtoTRAK Control models:
RMX and RLX

**TRAK MACHINE
TOOLS**



SOUTHWESTERN INDUSTRIES, INC.

2615 Homestead Place
Rancho Dominguez, CA 90220-5610 USA
T | 310.608.4422 | F | 310.764.2668
Service Department: 800.367.3165
e-mail: sales@trakmt.com | service@trakmt.com | web: www.trakmt.com

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TRAK Machine Tools
Southwestern Industries, Inc.
2615 Homestead Place
Rancho Dominguez, CA 90220
Phone 310-608-4422 ▪ Fax 310-764-2668
Service Department
Phone 800-367-3165 ▪ Fax 310-886-8029
Web: www.trakmt.com

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1.0 Introduction

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

With the ProtoTRAK RX DXF converter, you may quickly and easily create ProtoTRAK part programs using the data stored in a DXF or DWG file. The ProtoTRAK RX DXF Converter gives you advanced features such as layering, chaining, automatic gap closing and even the ability to modify the part drawing.

With the ProtoTRAK RX DXF Converter, you are in complete control of the creation of your 2D or 2 1/2 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 the part program in the way you direct. From there, you have access to the powerful Defaults and Options for generating the tool path you want.

1.1 DXF Versions

This version of DXF is 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-RMX	DXF converter for mills
CONV-DXF-RLX	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 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 2006. 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 these products should also work with this converter, as long as those files are saved with the DXF or DWG extension.
- The ProtoTRAK RX DXF Converter may be used with 2D drawings, it is not intended for 3D or Solid model drawings. We offer a Parasolid Converter product for that application.

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.

In addition to what is written above, the RX control has what is called a File Based Write Filter (FBWF) implemented. This protects the operating system and our software from malware. Malware includes computer viruses, Trojan horses, ransomware, spyware, etc.

The goal of the write filter is to enable a device to be configured a certain way and then, when it's rebooted, revert to a default clean state, with any writes from the last boot session wiped away.

The FBWF can be disabled via a service code so users can set up a network or commit a driver change (for example for a new USB memory device) to memory. It should be noted that when the FBWF is off, the operating system is vulnerable to malware so a user needs to be extra careful. Viruses that are now introduced will be there forever once the FBWF is turned back on. Our software will turn this FBWF back on automatically once you exit this service code.

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 mSATA memory device, 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 TRAK Machine Tools. 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 on your ProtoTRAK RX CNC by tapping the Status Info Key and looking under Options Activated.

2.1 Installing the DXF 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.trakmt.com.

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. Select 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 TRAK Machine Tools Sales Support at 800-421-6875. 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 TRAK Machine Tools Sales Support at 800-421-6875. After taking the serial numbers, Sales Support will give you your password.
5. After the password is entered, the DXF Converter is ready for use.

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.

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 the file type chosen, ACAD DXF or ACAD DWG. Press Open File and the DXF Converter will begin to process the file.

After the file opens, the ProtoTRAK RX will revert to the Start Screen. Tap Program to begin and navigate away from the start screen by swiping, or by tapping either Go To Begin or Go To End.

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.

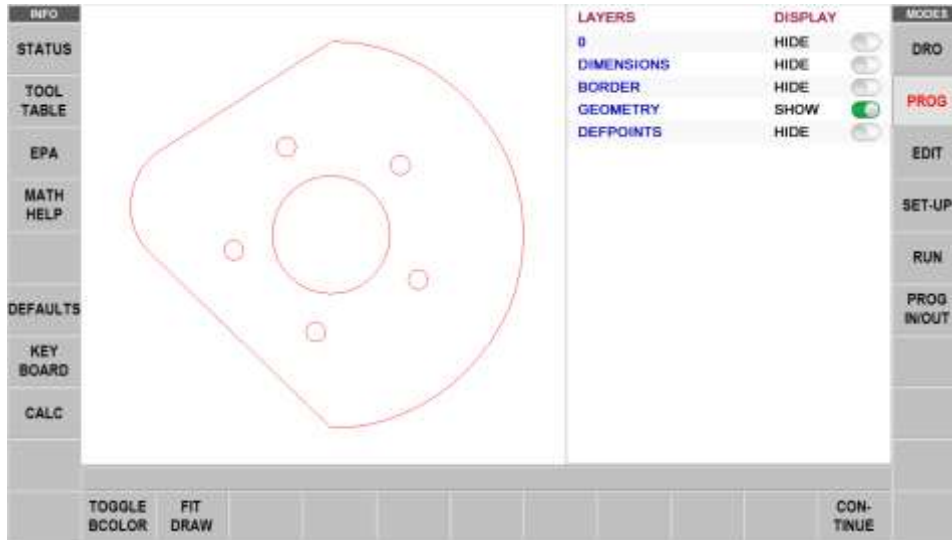


FIGURE 3.2.2 Only the Geometry layer is shown.

You can go between Show and Hide as often as you like as you choose which layers to eliminate from view.

TOGGLE BCOLOR – Toggle between colors to show a white or black background.

FIT DRAW – Size the remaining Geometry layer to fill the screen (as we have done in the figure above).

CONTINUE – Start programming the part.

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 to the Gap Tolerance, 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 such as shown below:

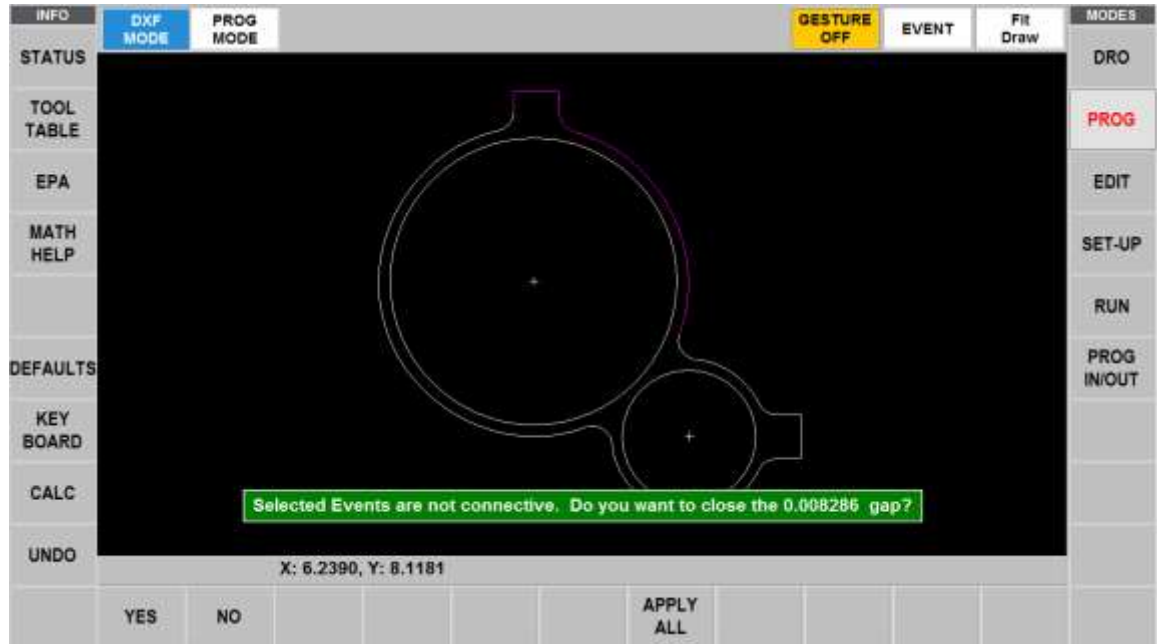


FIGURE 3.3.2 A gap is encountered that exceeds the gap tolerance.

- 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 select 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 Aligning the Part

In order to make the part easier to program and locate on the machine, you may want to align the part features differently than they are aligned on the drawing.

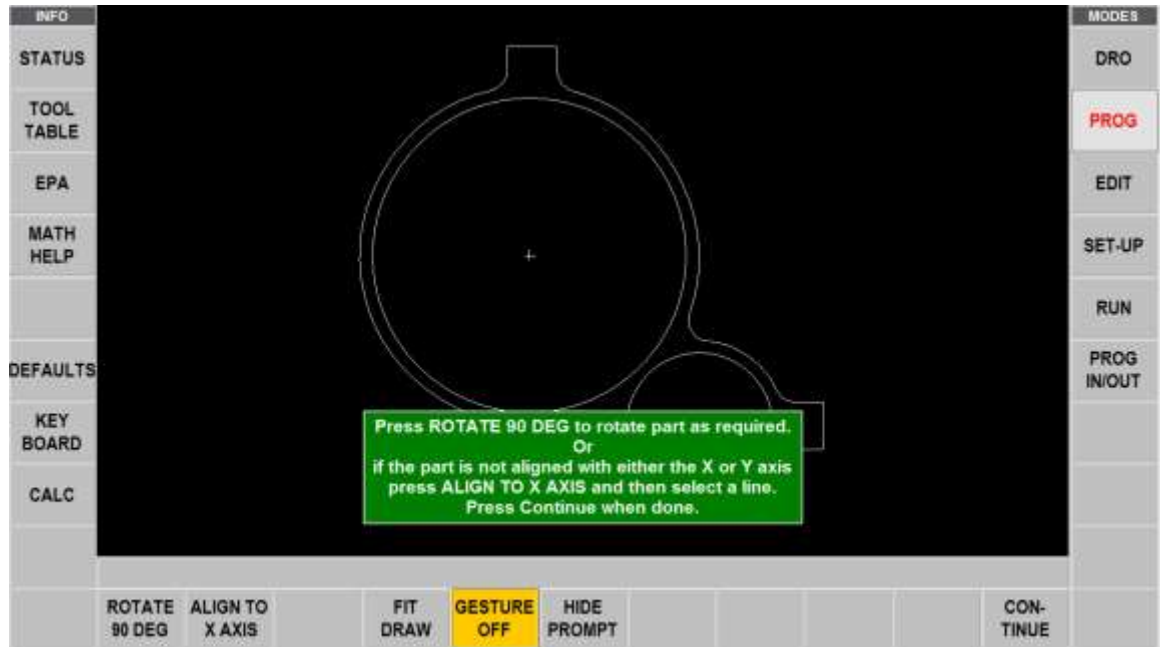


FIGURE 3.4 Aligning the part drawing

You may use the touchscreen Gestures to move the part. In addition, the following soft keys are active:

ROTATE 90 DEG – Will rotate the part 90 Degrees with each tap.

ALIGN TO X AXIS – Will prompt you to identify the line that you want aligned on the part drawing. Select the line you want to align and then press the line a second time for the alignment to be applied.

FIT DRAW – Will size or resize the drawing to the screen (especially helpful after zooming or panning).

GESTURE OFF – Will disable the touchscreen gestures. This may be useful if you find yourself trying to select a line but instead the screen pans. Press this button to turn off gestures. Press it one more time to turn gestures back on.

The following Gestures are used in the DXF software when using it on a non-touchscreen desktop computer. Please note that pan and zoom work differently between DXF and Program mode within DXF.

DXF Mode

- Zoom: Click on hold the right mouse button, and then slide the mouse up to zoom out or down to zoom in.
- Pan: Click and hold the left mouse button, and the slide the mouse in any direction that you wish to move the drawing. You can also pan up and down by using the scroll wheel on your mouse.

Program Mode

- Zoom: Use the scroll wheel on your mouse, scroll up to zoom in or scroll down to zoom out.
- Pan: Click and hold the right mouse button, and then slide part in direction you want to move the drawing.
- Rotate: Click and hold the left mouse button, and then slide mouse in any direction you wish to rotate your part three dimensionally.

HIDE PROMPT – Will toggle the green message box in case you want to access the drawing underneath it.

3.5 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.

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 select the two pieces of geometry that will intersect at the point you want to use. After you select 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.

Choice B: An End Point

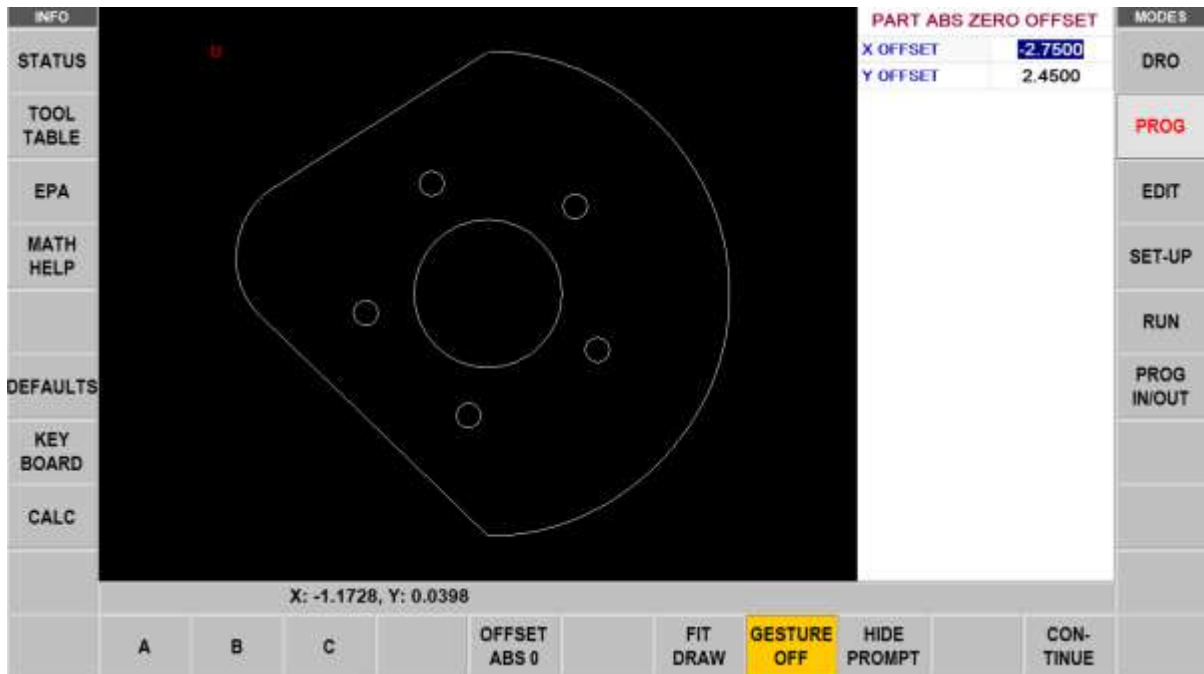
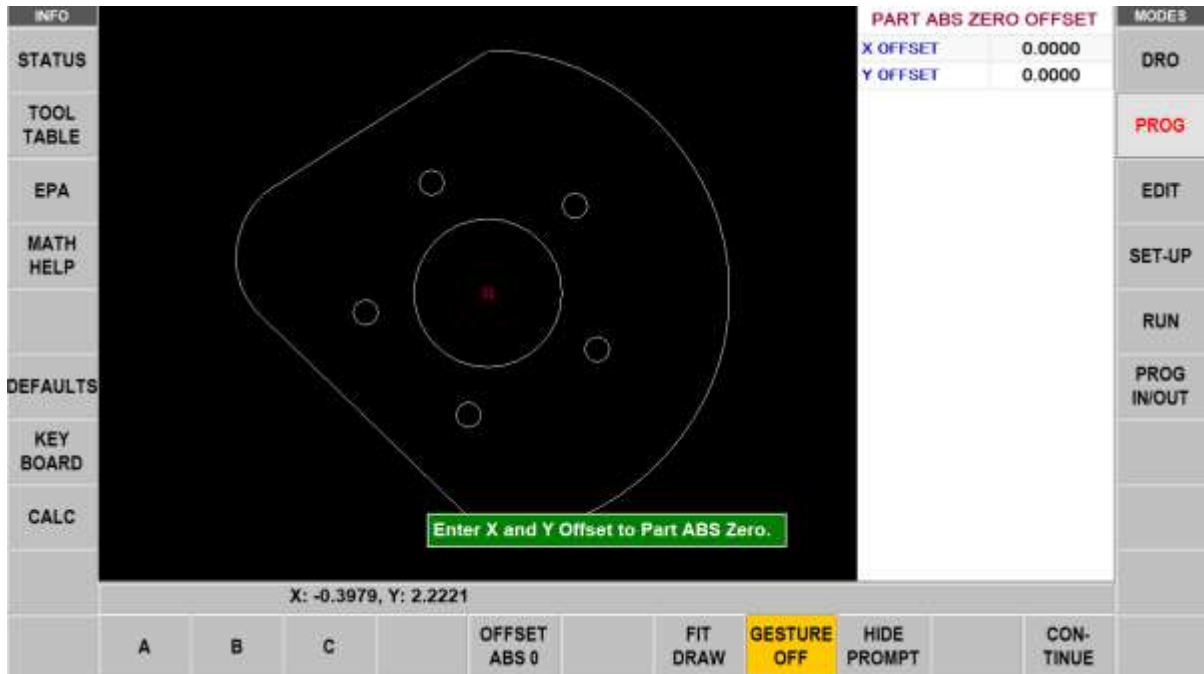
This allows you to use an end point of a line or an arc. Select a line or arc close to the end point to be used as the absolute zero reference. Circles cannot be used with this choice.

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.

Offset Abs Zero:

This allows you to offset the zero point chosen by some offset. Fill in the X and Y offset prompts as shown in pictures below to offset your zero from the location chosen.



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.

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

3.6 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. Select 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, select 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 tapping (or clicking) on the screen. Select 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, select 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.

Hide Line: Click hide line then click on an unwanted line to hide it. Click UNDO to bring it back.

Break Geom: Allows intersecting lines to be broken into separate line segments. Click on one segment of one of the lines and then on one segment on the other line. Click on UNDO to unsegmentize the lines.

Inquire Geom: Click key then click any line, arc or circle. It will open a window that shows the dimensional data for the geometry you selected. When clicking a circle, the diameter and center points of the circle will be shown. This feature may be useful to determine if you need to break geometry apart to create your part.

UNDO – Eliminates the last entry or item. This key only appears when there is something that can be undone.

3.7 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.

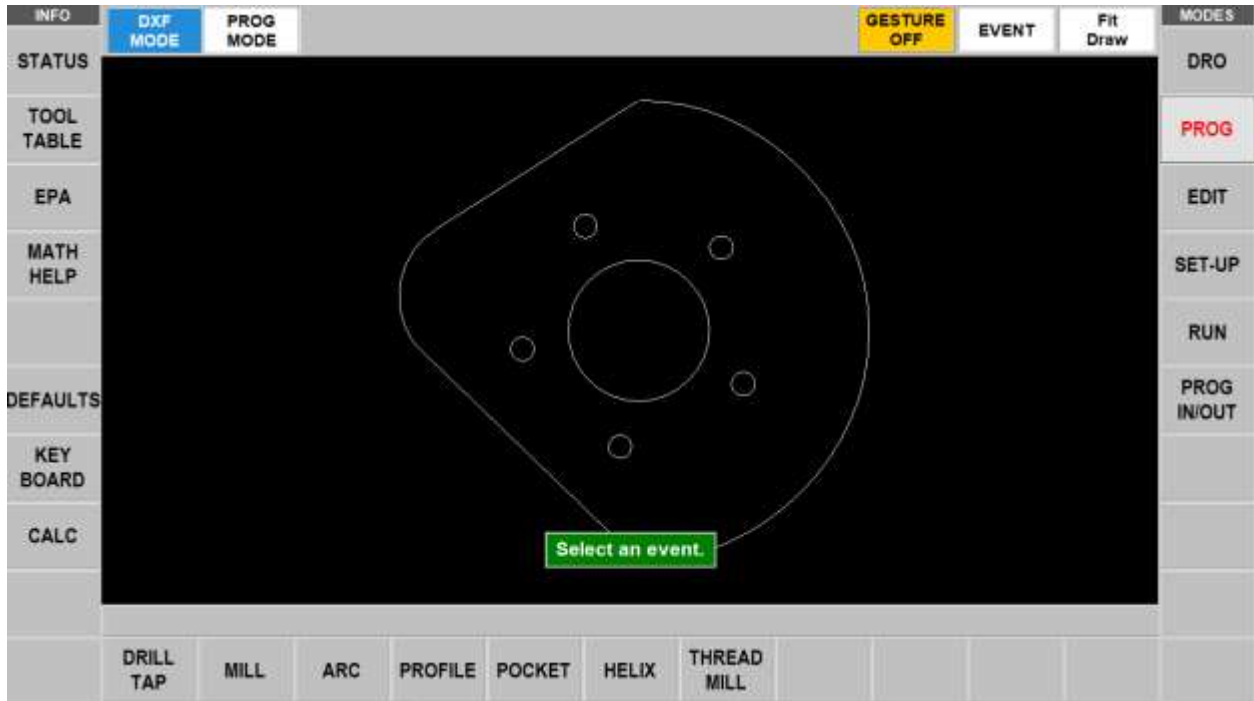


FIGURE 3.7 The part is ready for us to define geometries.

3.7.1 Event Types

Drill Tap: Allows the conversion of any circle into a Drill, Tap or Bore event. You should select all the similar holes at the same time. For example, we would select all five of the holes in the bolt hole pattern in Figure 3.7 above at the same time. That way, when we select EVENT and fill in the prompts for the first one, the other five will be programmed with the same information.

Mill: Allows the conversion of any straight line into a Mill event. Select 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. Select 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.

Helix: Allows the conversion of a circle into a Helix event.

Thread Mill: Allows the conversion of a circle into a Thread Mill event.

Island: Allows the conversion of any connecting lines and arcs into an Island event. When selecting an Island event, the first two lines clicked determine the direction of the profile of the Island (Clockwise or Counterclockwise). Island segment geometry

must be connected to each other and must be closed – it needs to end where it began.

3.7.2 Functions Active in This Screen

Undo: The undo feature is active in this screen. Tapping this will undo one step for each time it is tapped while still in the same event group (event groups are described in section 3.7.3 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: Tapping the Event button will change the screen to show 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.

Tap Event again to return to the Select and Event screen.

DXF MODE: Select this key to be in the process of using the DXF file to select the geometries for programming.

PROG MODE: Select this key to go into the Program Mode (within the DXF process). You would want to do this if you wanted to program something that was not on the drawing file, for example a Face Mill to clean up the stock or a Position move to clear a clamp.

FIT DRAW: Will resize the drawing to fit on the screen. This would be necessary if you had zoomed or panned the drawing in order to pick a geometry.

3.7.3 Event Groups

You create event groups out of items on the drawing that you want to machine the same way. Select 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 select 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 select 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 select 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, select the drill softkey again, or select something

other than Drill, and then select Drill again. You will now be working on a new event group.

3.7.4 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 select 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 unselect any number of steps in the chain, one step at a time.

3.7.5 Color Schemes and Status Flags

Throughout 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 select 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.

3.8 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.

4.1 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.

4.2 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.

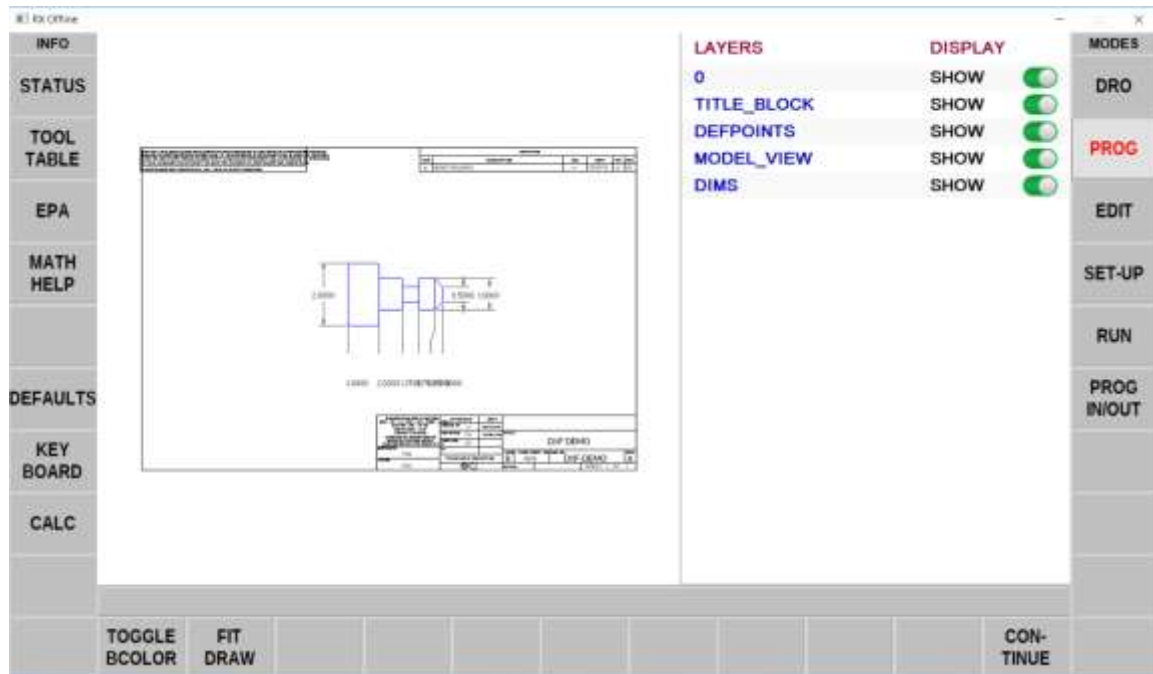


FIGURE 4.2.1 The first screen you see when the DXF Converter launches. Notice that the entire drawing, including borders and dimensions, is shown on this first view.

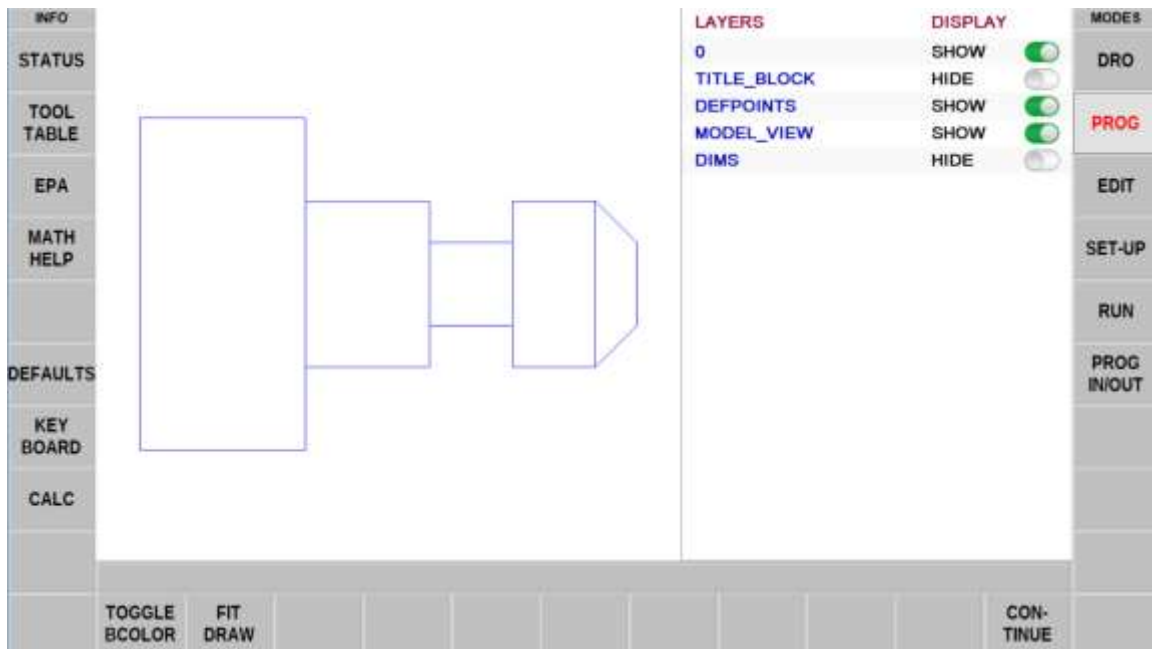


FIGURE 4.2.2 Through a process of elimination, we have isolated the view of what we want to machine.

You can go between Show and Hide as often as you like as you choose which layers to eliminate from view.

TOGGLE BCOLOR – Toggle between colors to show a white or black background.

FIT DRAW – Size the remaining Geometry layer to fill the screen (as we have done in the figure above).

CONTINUE – Start programming the part.

4.3 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.

4.3.1 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 cycle.

4.3.2 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 the message: "Events are not connective. Do you want to close the 0.050 gap?" Select Yes or No and continue.

4.3.3 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.

4.4 Rotate Drawing and Pick a Zero Absolute Reference

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.

If the drawing is skewed (this is rare), tap Align to X Axis to correct it.

***Note:** All gestures that you use in creating ProtoTRAK programs are also available in DXF programming.*

4.5 Setting an 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.

4.5.1 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 tap on the two pieces of geometry that will intersect at the point you want to use. After you tap the second piece of geometry, a red x will appear to show you where the absolute zero is now set.

4.5.2 Choice B: Closest Endpoint of Line or Arc

This allows you to use an end point of a line or an arc. Tap a line or arc close to the end point to be used as the absolute zero reference.

4.5.3 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 tapping the line or arc.

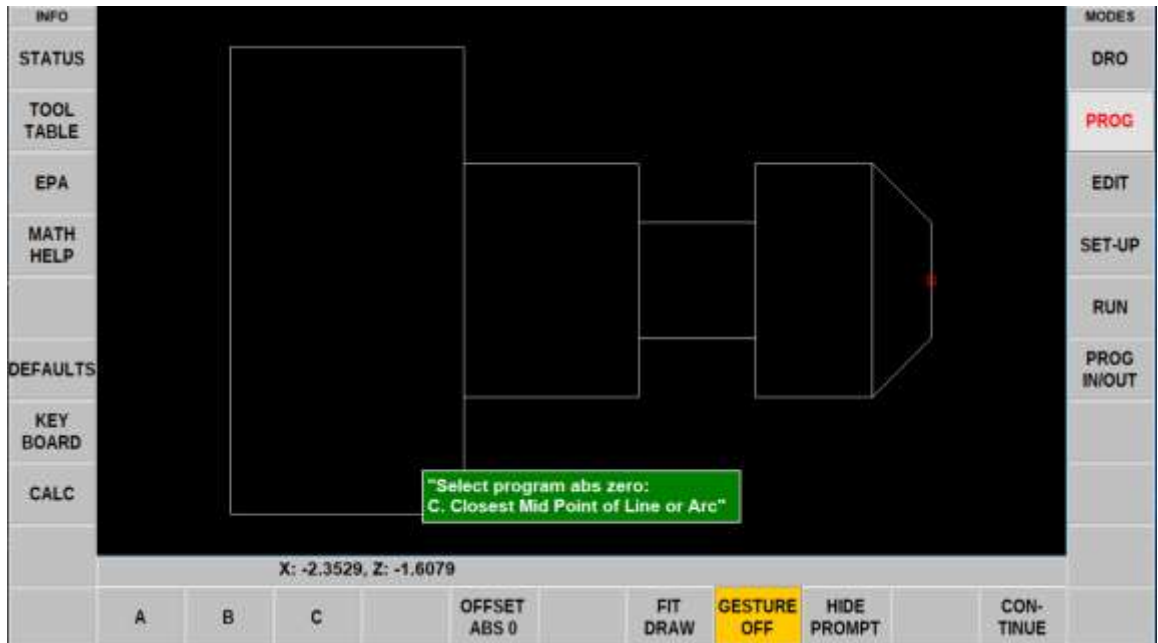


FIGURE 4.5.3 For this exercise, our best option is to go with C, Midpoint of line or arc and tap the line at the very front of the drawing.

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

4.6 Positive Half, Negative Half and Full Drawing Display

The ProtoTRAK DXF Lathe converter allows you to display the Positive X portion 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 key will display the portion of the drawing from the ABS 0 point you specified, down. This option is generally best.

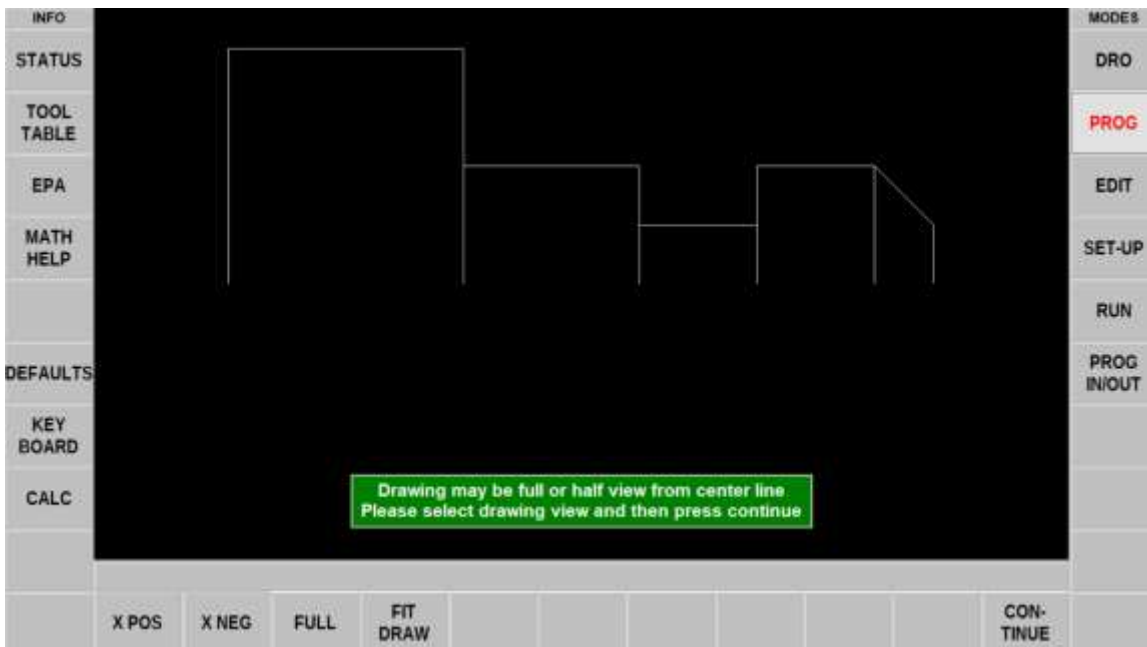


FIGURE 4.6.2 The X neg key will display the portion of the drawing from the ABS 0 point you specified, up.

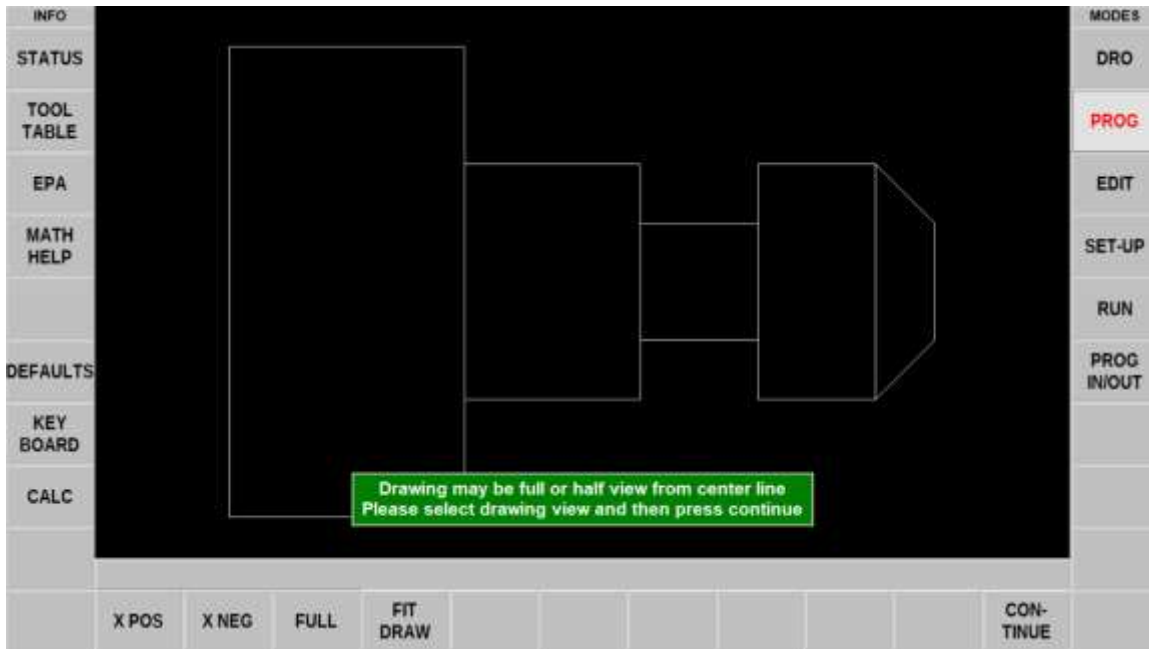


FIGURE 4.6.3 The FULL key will restore the drawing to its entirety.

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.

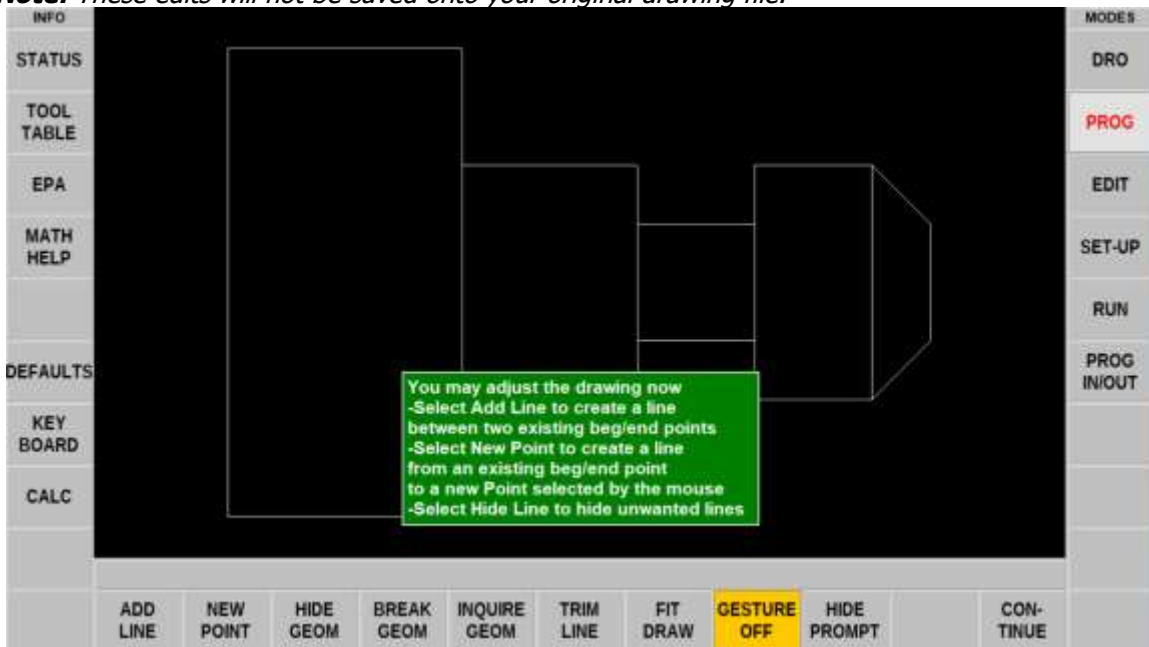


FIGURE 4.7 The Edit Screen

4.7.1 Add Line

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

To add a line using the Add Line feature, first tap Add Line, then tap the begin or end point on each line. See figures 4.7.4a and 4.7.4b.

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

4.7.2 New Point

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

To add a line using the New Point feature first tap on New Point, then tap 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 your finger. Tap where you want it to end.

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

4.7.3 Hide Line

The Hide Line feature allows you to hide a line as created in the DXF file. This is helpful when using the Chain function (explained in section 4.8.2) on a cycle event.

To use the Hide Line feature, tap Hide Line, then tap the line you would like to hide, this will highlight the entire line purple. Tap a second time to hide it.

4.7.4 Trim Line

Trim Line works the same as Hide Line but only hides the portion of a line that doesn't cross another line.

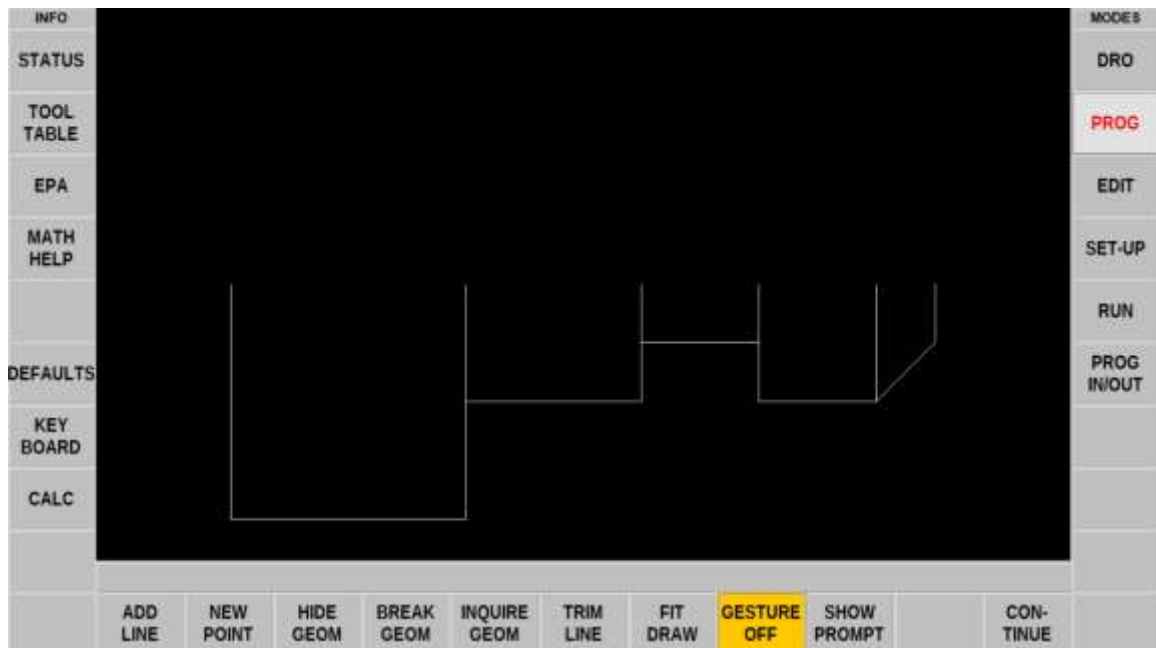


FIGURE 4.7.4a Drawing of an unedited part

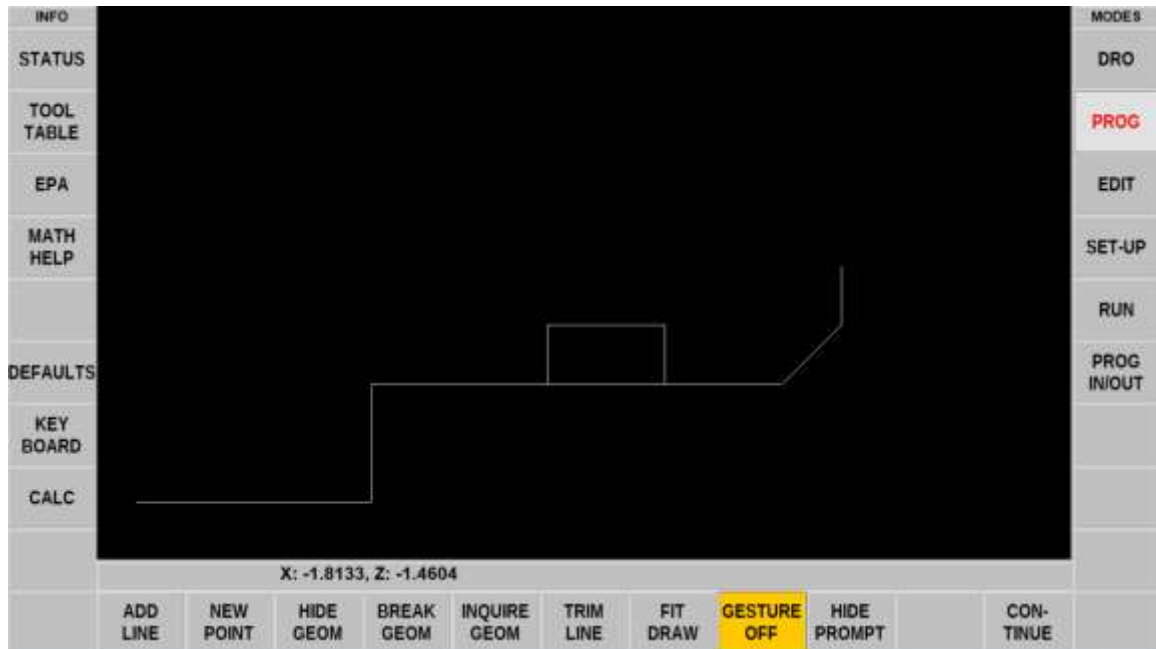


FIGURE 4.7.4b Drawing of part with an Add Line across the groove (allows you to program a cycle excluding the groove) and has Trimmed Lines that do not represent tool path.

4.7.5 Break Geometry

This feature allows you to create two separate lines out of one at a point where a second line intersects it. Tap Break Geom, then tap the line you want to break into two and finally tap the line intersecting the line you want to break. The DXF data will now recognize that each of the two separate lines has their own begin and end points and can be selected on their own.

4.7.6 Inquire Geometry

This feature allows you to identify dimensional information on the drawing. To use it, tap Inquire Geom and:

- Tap a line to see its begin and end points as well as its length
- Tap an arc to see its begin, end and center points as well as its radius
- Tap a point to see its location



FIGURE 4.7.6 *Inquired geometry for a line*

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.

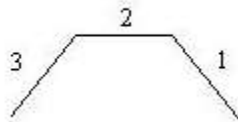
Turn: Converts any straight line into a Turn event. Click on the end of the line where you want to establish the beginning point (X Beg, Y Beg).

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).

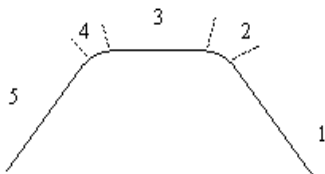
Cycle: Converts connecting lines and arcs into a Cycle event. When selecting a Cycle event, the first line clicked determines where the Cycle starts and the second line clicked determines the direction of the Cycle. The cycle event inside of the DXF converter (only) has an X stock prompt. This field 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.

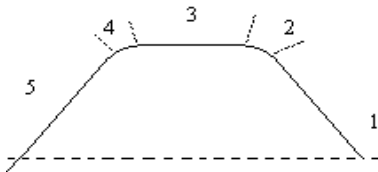
Groove: Allows the conversion of three lines, in a specific order, into a groove event.



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 started. 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 4.3.3).
- An intersection or branch that requires a decision from you. If this is encountered, the system will prompt you to select the direction by tapping the next line or arc segment you want included in the cycle 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.

4.8.3 Color Schemes and Status Flags

Throughout 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.

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.

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

When you tap 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 tap 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, tap 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, tap the Event button, and complete the event.

4.8.5 Programming in the Normal ProtoTRAK Mode

In the Select Event screen there are two keys in the upper left part of the screen labelled DXF MODE and PROG MODE. Normally you would program a DXF part by selecting geometry from the DXF drawing and then completing the event. This is the DXF MODE as it has been described in this manual.

In addition, you may tap PROG MODE at any time and program events exactly as you would program without DXF. These events will be part of your final program when and where you create them. However, they will not be back driven into the DXF drawing. Even if you are programming exclusively in DXF you may want to occasionally tap PROG MODE to see what you have programmed so far.

4.9 Creating a DXF File from a ProtoTRAK

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.

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!)

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.

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.