

ProtoTRAK®

KMX Offline Programming Manual

For CNC models:

- ProtoTRAK KMX

Includes DXF, Parasolid & Verify Options

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TRAK MACHINE
TOOLS



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TRAK Warranty Policy	Error! Bookmark not defined.

1.0 Introduction

The ProtoTRAK Offline Programming System allows you to write, edit, and inspect programs for your ProtoTRAK using a standard PC computer.

Programming with the Offline system is essentially the same as programming through the ProtoTRAK keyboard with a few differences. This manual will provide installation instructions for the offline and highlight the differences in programming between using the offline and the control pendant. The basic programming instruction is left to the programming, operating, and care manual you received with your ProtoTRAK machine.

The KMX Offline can also be ordered with an option that allows the user to bring in DXF and Parasolid files and convert them into ProtoTRAK part programs. This option also contains Verify software that allows a user to simulate the tool path prior to running the part. This option is ordered under OL-CONV.

This manual is a part of the Offline Programming option(s) ordered with part numbers PTKMXOL and OL-CONV. The offline programming option includes:

- KMX Offline USB Option key (P/N 23770-3) or KMX Offline plus DXF, Parasolid and Verify with USB Option Key (P/N 23770-4)
- Manual (P/N 29921)

In addition, you are entitled to a free copy of each of these programming manuals, download from website www.trakmt.com

2.0 Offline Computer Requirements

The ProtoTRAK Offline is designed to run on an IBM compatible PC, Pentium II or higher, 256 MB RAM, 100MB free hard disk space, and Windows 7 or later.

The software also requires Microsoft .NET Framework 4.5, as well as Microsoft Visual C++ 2008 library, which will be part of the download. In order to save programs or use software options, it will also be necessary to install an option key in the USB port (see Section 3.1), as well as the drivers for it.

3.0 Installation

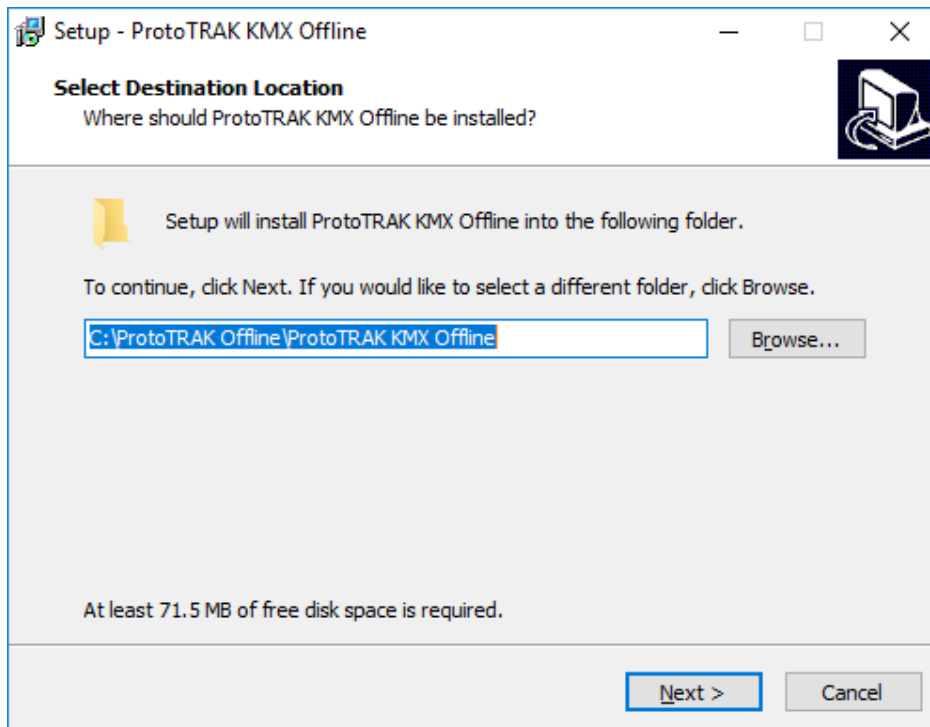
You can download our software from www.trakmt.com.

Navigate to the RESOURCES & SUPPORT option, and then click on SOFTWARE.

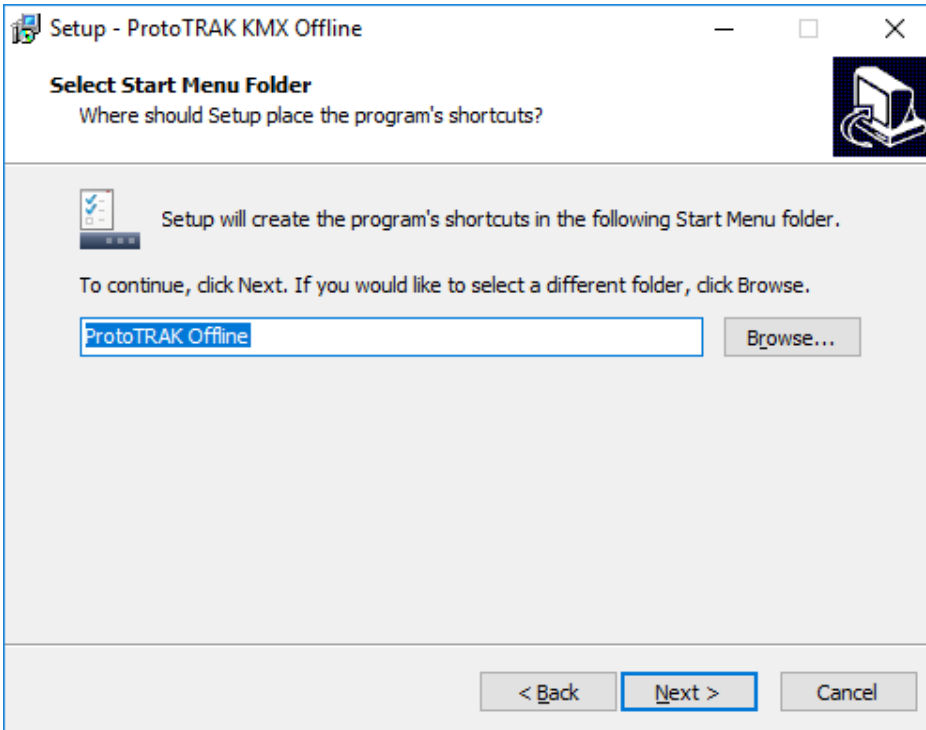
Find the ProtoTRAK KMX Software, and click on the link to take you to the download and instructions.

3.1 Installing the ProtoTRAK KMX Offline Software

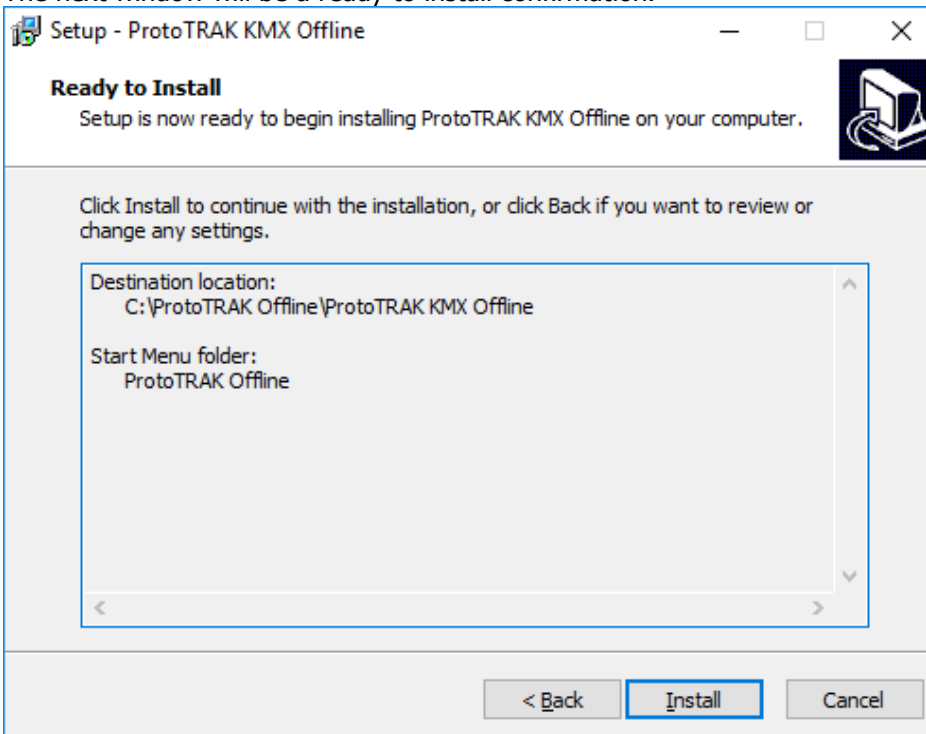
After running the downloaded file, the first window may look as follows, prompting you for the location for which you'd like to install.



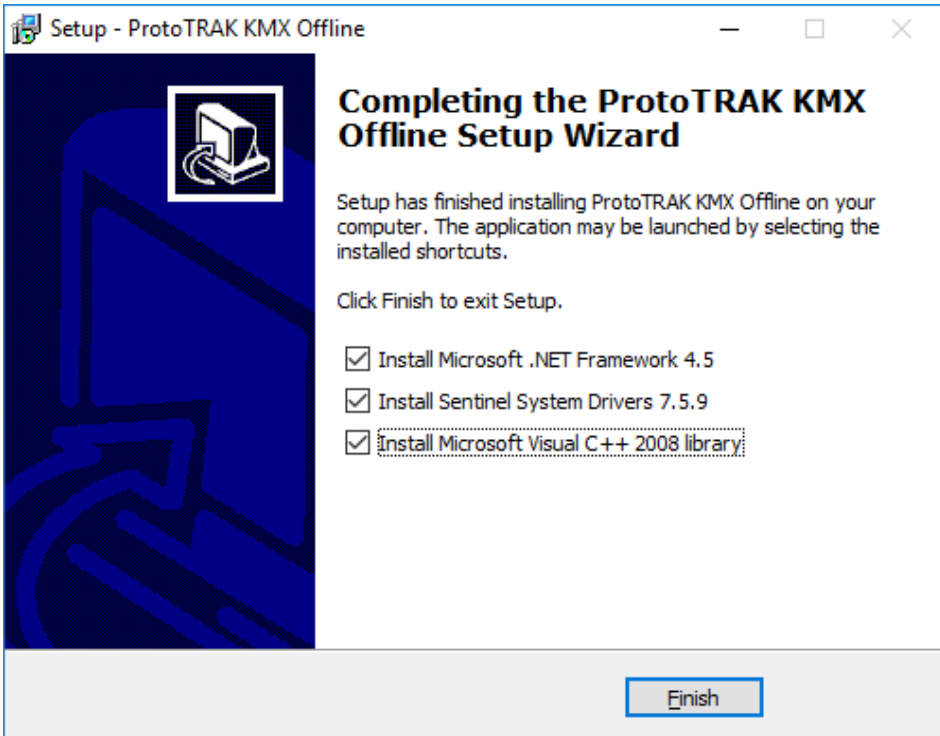
Next it will prompt you for the Start menu shortcut names.



Note that if you already have a previous version of software installed, then it will skip these first two windows and simply overwrite the previous installation with the latest version. The next window will be a ready to install confirmation.



After you click on Install the setup will copy the necessary files for the KMX software onto your computer. The last window will then prompt you for the additional required files.



All items that have a check mark activated will be installed after the Finish button is clicked. Note that all 3 items must be installed, as they are required for the KMX Offline software to function properly. However they only need to be installed once, so if you are simply updating your offline software to the latest version, you can uncheck them and continue on.

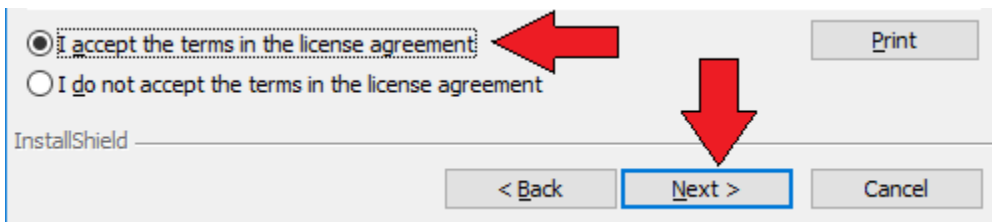
3.2 Installing additional requirements

3.2.1 Sentinel System Driver

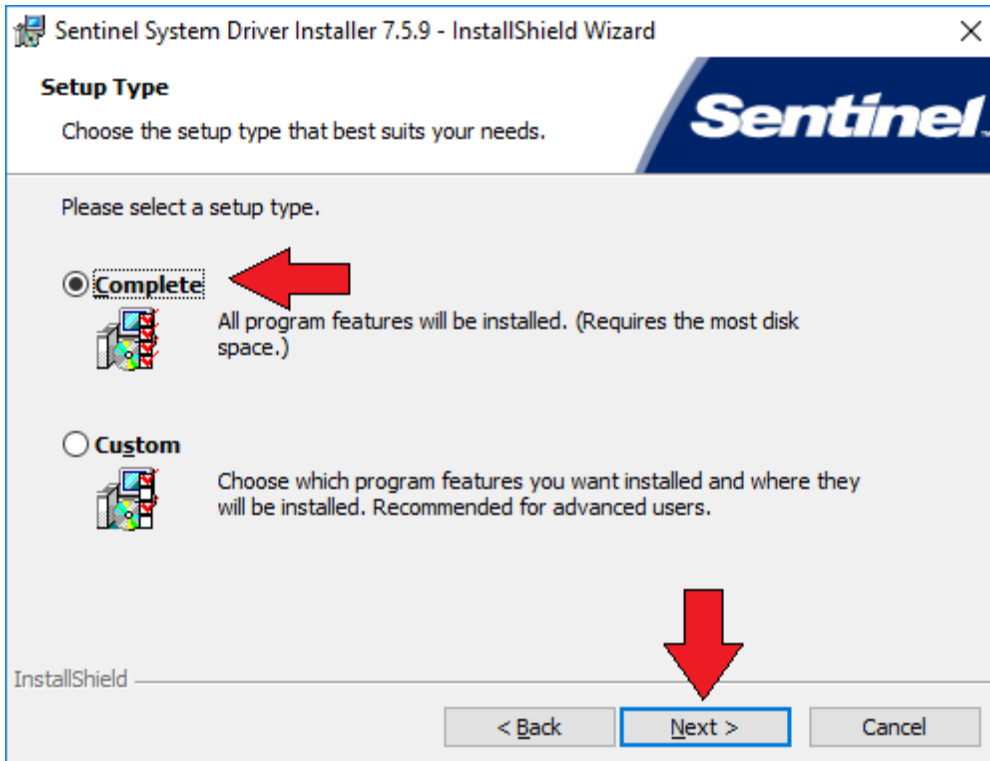
The Sentinel System Driver is required in order for you options key to be recognized on your PC properly. When installing this, you will see something like the picture below:



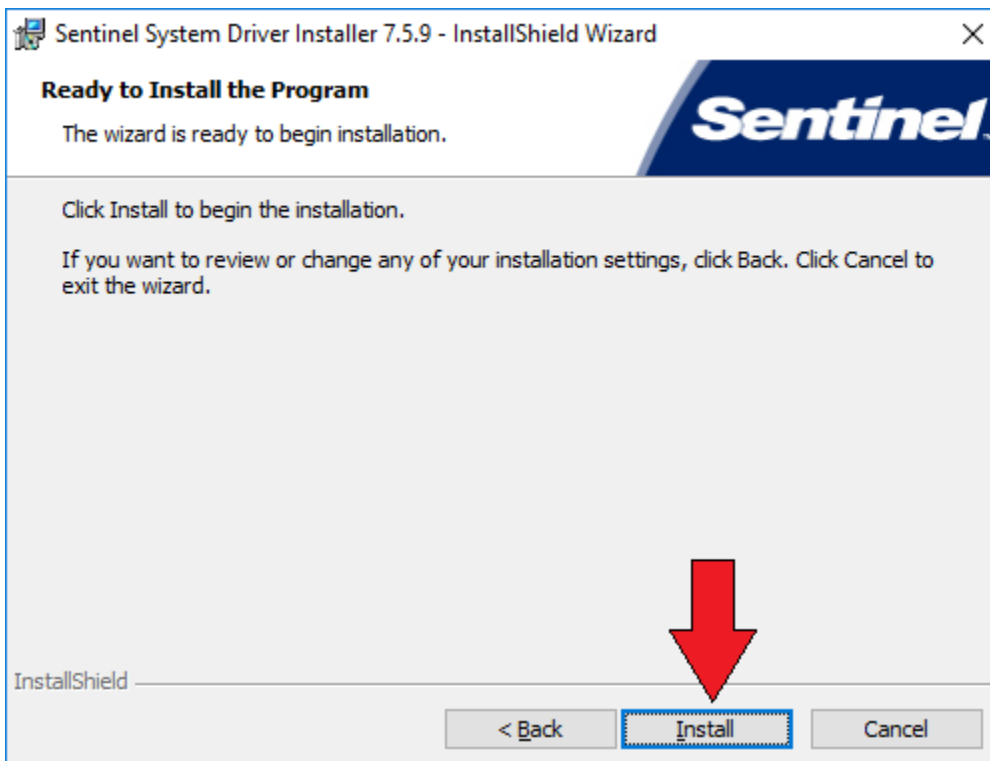
Simply click on the Next button to proceed with installation.



Click on the "I Accept" selection and then the Next button once again.



If prompted, select the Complete installation, followed but the Next button.



Then click on the Install button to proceed with the installation.



Once the wizard has properly installed the necessary software, you should see a confirmation, and then click on the Finish button to complete.

Note – if the installation prompts you to restart your computer, select NO so that you can continue with the rest of the installation and restart at a later time.

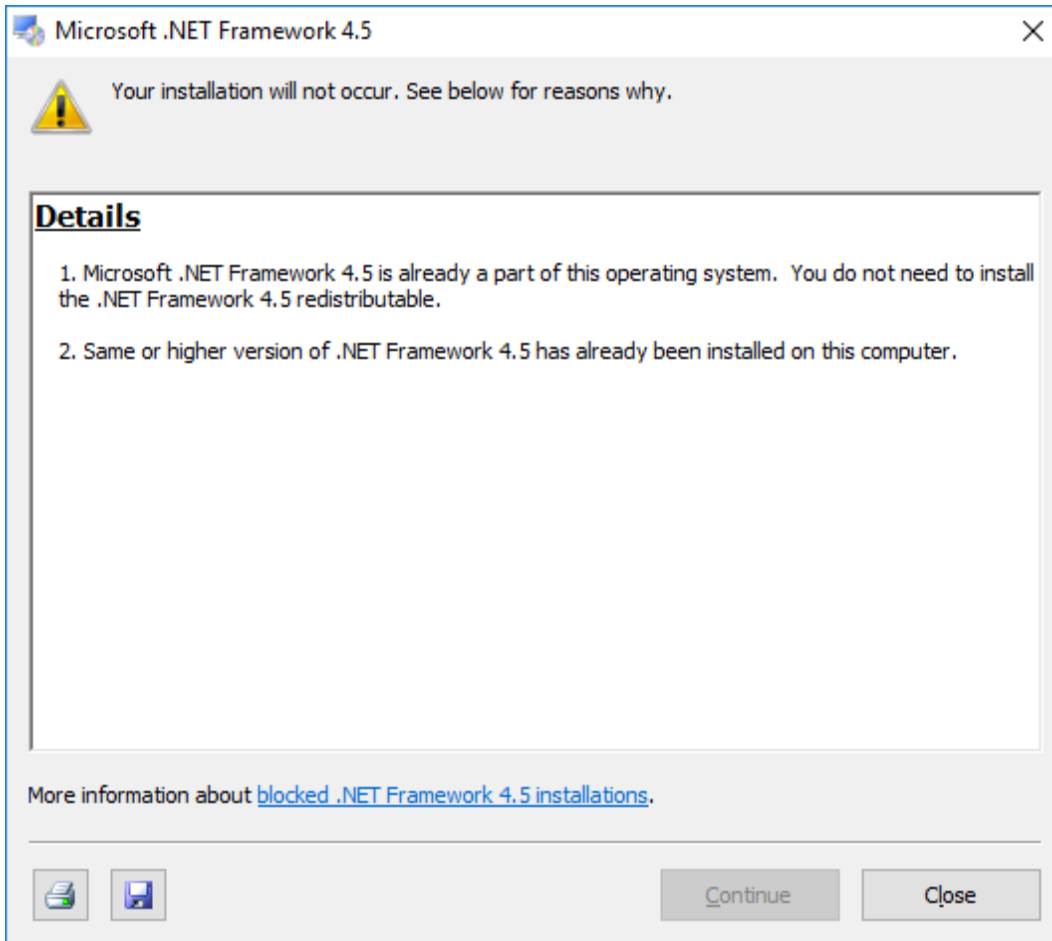
In the event that you already have the sentinel drivers installed, or have an older version present, you will get a message indicating this. You can choose to either Upgrade if some older drivers are detected, or Close the installation if the drivers are already present.

3.2.2 Microsoft .NET Framework 4.5 Installation

The next installation is necessary in order for the Run Panel to function properly. You need to have at least version 4.5 of .NET framework installed on your computer.

Simply follow the steps accordingly, and note that the installer may need to download additional files from the internet to your computer.

In the event that the software is already installed on your computer, you will see a window notification.



Simply click Close to continue.

3.2.3 Microsoft Visual C++ 2008 distribution

This installation should run silently in the background. When prompted to install, it will not ask for any user input and should install all the necessary files within a matter of seconds.

3.3 Installing the USB Options Key



Note: The purpose of the Sentinel Drivers is for the USB dongle you'll have obtained for your offline. The drivers are a set of instructions for the computer on how to interpret the functionality of the dongle. The dongle itself serves as a key, enabling your offline to perform most if not all the functions that it's capable of. Without this, your offline will simply work as a demo and you'll be unable to save files and perform other functions. This dongle contains the options you've purchased, such as the DXF, Parasolid and Verify option.

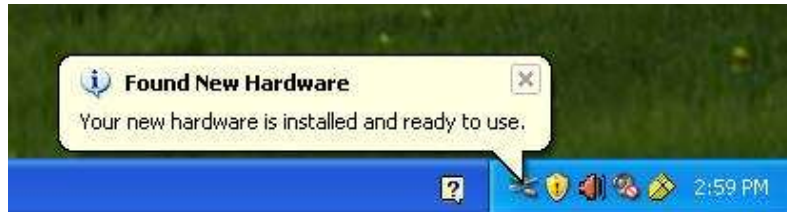
Before plugging the USB option key into your PC, please ensure that you have installed the software as described in section 3.2.1.

1. If you have not already done so, go ahead and restart your computer as you were previously advised to do so.

2. Once computer has restarted, insert dongle into an open USB port.
3. A USB icon should appear near the time with a bubble floating above it reading "Safe Net USB UltraPRO/SuperPro".



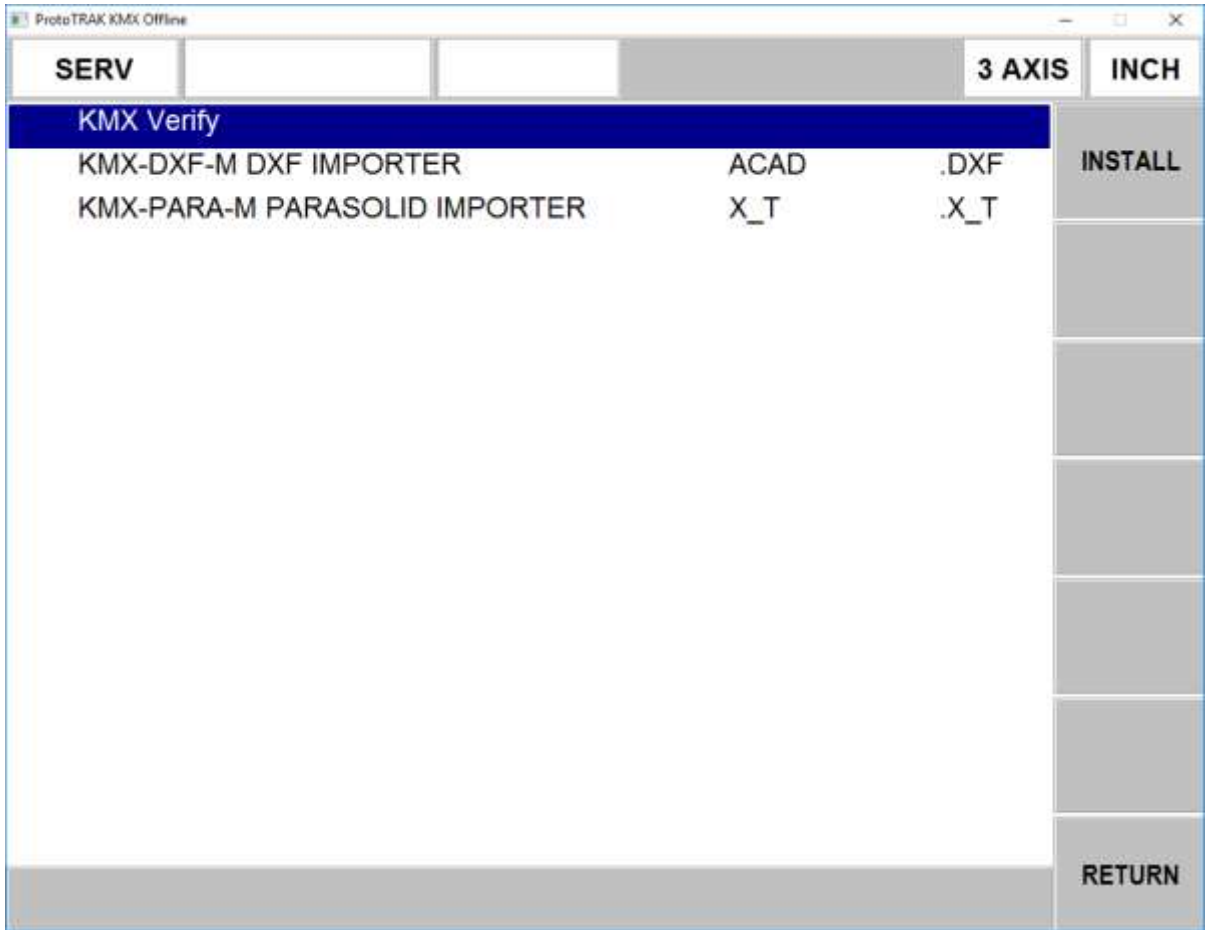
The "Found New Hardware" bubble will then follow. Once it disappears, the dongle should be fully installed.



3.4 Checking the Option Key Installation

To check to ensure that the option key is installed properly, you can open our software and do the following:

1. Launch our KMX Offline Software
2. Go into SETUP and then SERV CODES.
3. At SERVICE CODE #, type in 318, and then press ENTER.
4. The next screen (shown below) should show you a list of options that you have purchased if the option key is properly recognized.
5. If the options are turned on, then will be in black letters and no further action is required. If the options are in gray, this means the options are not turned on.
6. Press the install button after you highlight the option you require. 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.
7. If you do not already have the activation password, write down the Hardware Key and Converter serial numbers shown on the screen for each option and call SWI Customer Support. After taking the serial numbers, SWI Customer Support will give you your password. Please note, these options are not free. If you have not placed an order for the KMX offline converter options (DXF, Parasolid & Verify), you will need to do so before we can give you the activation password.
8. Please note that 3 separate passwords will be required to turn each of the options on.

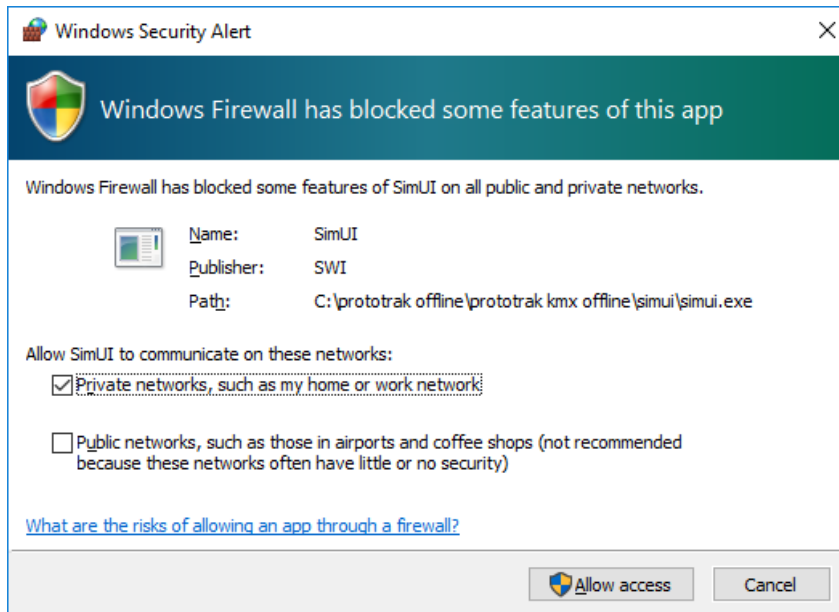


Otherwise, it will display an error informing you that the hardware key is not recognized. If you have followed the instructions and continue to have problems, contact the Service Department at (800) 367-3165 for further assistance.

4.0 Programming with the ProtoTRAK Offline

4.1 Getting Started

Once you launched the offline software for the first time, you may see a security alert window that looks similar to below:



You can click on the “Private Networks” selection, and then click on “Allow Access” to continue. This is only to allow the Run Panel full access to our software, and will not be communicating with anything else outside of that.



Once our KMX Offline Software launches, it should look similar to the picture above.

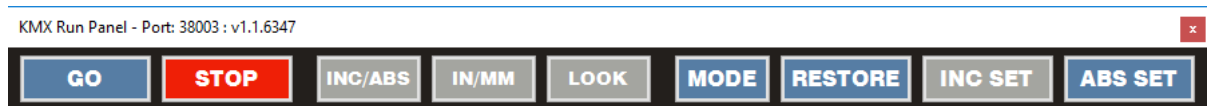
Note that the offline software includes the ability to program in both 2 axis and 3 axis modes. To toggle between them, simply click on where it says "GO TO 2 AXIS" or "GO TO 3 AXIS" to take you to each mode respectively. To get back to this screen at any point, simply press the MODE button, followed by ALT + B to go back one screen.

4.2 ProtoTRAK KMX Run Panel and Keyboard Equivalents

Below is a view of the KMX front panel.



The KMX Offline software introduces a Run Panel provides the user with access to the most commonly used front panel buttons used in programming his parts.



You can click on any of the buttons displayed and they will react with the software exactly as they would on the machine itself.

You can also right click on the title bar of the Run Panel window and select the option to rotate it if you prefer.



The Run Panel is designed to always be on top. If you wish to hide it, you can click on the icon in the Windows taskbar, and then click on it again to restore it.



In addition, the KMX Offline Software still supports keyboard shortcuts if you prefer. They are as follows:

ProtoTRAK KMX Key	Keyboard Equivalent
Right column of soft keys	F1 thru F8
GO	Alt + G
STOP	Alt + S
INC SET	Alt + I
ABS SET	Alt + A or Enter
X	X
Y	Y
Z	Z
MODE	Alt + M
BACK	Alt + B

ProtoTRAK KMX Key	Keyboard Equivalent
+/-	+ and -, default is positive
Numbers 0 - 9	Numbers 0 - 9
INC/ABS	Alt + [
IN/MM	Alt +]
LOOK	Alt + L
RESTORE	Use back/clear key
▲	Alt + up arrow
▼	Alt + down arrow
RESTORE	Alt + Backspace

4.3 Program Compatibility

While the KMX Offline may only have a limited amount of events available to program, it will still support other events that are programmed from other ProtoTRAK controls. If they are opened on the KMX, you will be able to see the event and edit them accordingly, however you will not be able to add new events. Such events include engraving and island pockets.

However some features may be ignored or generate an error. For example, the KMX does not support spindle speeds, and so if a program is opened that contains spindle speeds, they will simply be ignored. If a Tap event were to exist though, the software will generate an error, because it is not capable of running such an event.

4.4 Supported Screen Resolutions

The KMX normally runs at a resolution of 1280 x 800. However, in order to support common laptop screens of 1366 x 768, the software will automatically shrink itself down to support this resolution if detected as your primary monitor.

4.5 Opening / Saving Programs using the KMX offline software

The KMX offline software looks at the first available removable drive to open and save programs from, just like the actual machine. On that removable drive, there should be a folder called "KMX PROGRAMS", where all the part programs will be located. So, for example, if your USB flash drive was recognized as drive E on your PC, then the location of your programs would be E:\KMX PROGRAMS.

Please note that the KMX Offline software only recognizes files that have a numerical filename 8 digits or less, and must have one of the following extensions:

MX2	Legacy 2 axis mill
MX3	Legacy 3 axis mill
PT4	Native file format for KMX, supports 2 and 3 axis mill
CAM	Opens a gcode program to be converted into events
GCD	Opens a gcode program to be run as is
DXF	Opens a 2D DXF program*
X_T	Parasolid text file*

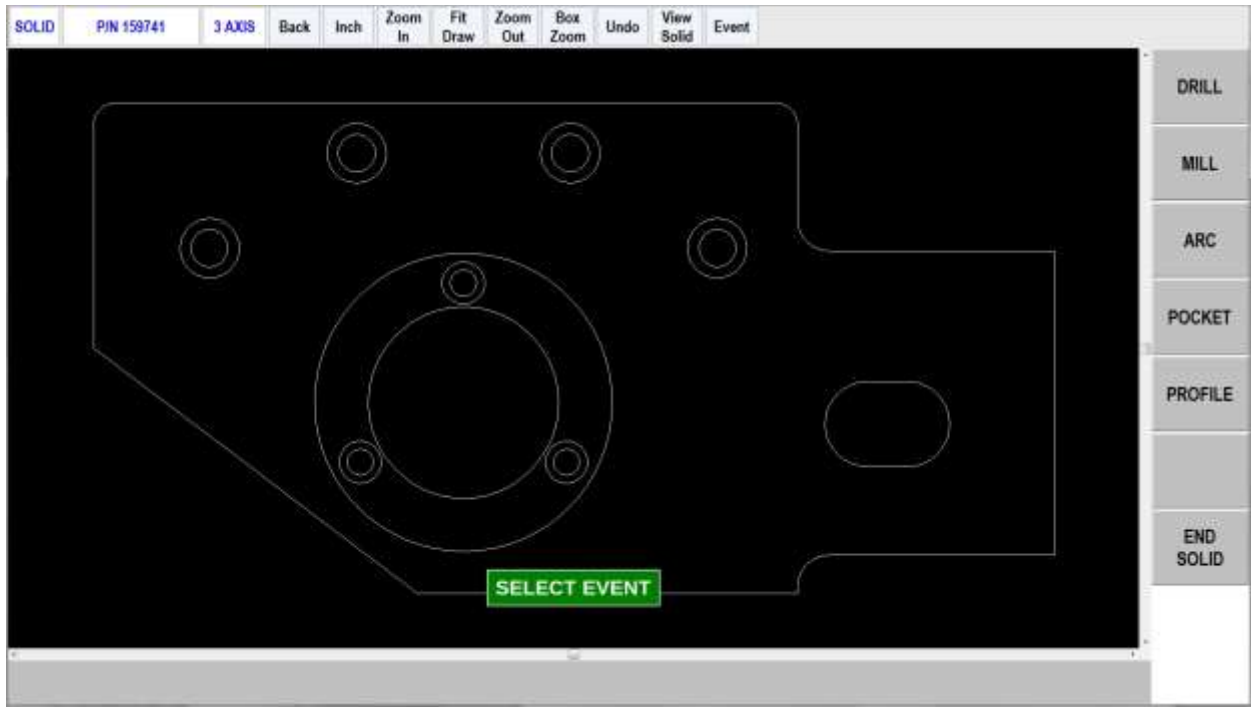
(* indicates optional feature)

Once the programs are saved to the flash drive, it is ready to take over to the ProtoTRAK KMX machine and once it is plugged in, the KMX control will be able to open them directly from the flash drive.

Also note that the KMX machine pendant will not accept DXF or X_T files, those must be converted on the offline software before taking over to the machine.

4.6 Resizing of DXF and Parasolid Screen

On both DXF and Parasolid software, you can grab the lower right hand corner of the screen to make the software larger on your screen. This may benefit those that have a high resolution computer screen. The picture below shows the screen expanded.



5.0 ProtoTRAK DXF Converter

5.1 Introduction to the ProtoTRAK DXF Converter

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.

5.1.1 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.

5.1.2 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 or 2002. 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 DXF Converter may be used with 2D drawings, it is not intended for 3D or Solid model drawings.

5.2 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 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.

5.2.1 Opening a DXF Part Program

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

Note: You can use Mode (or Alt + M on keyboard) 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.

5.2.2 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.

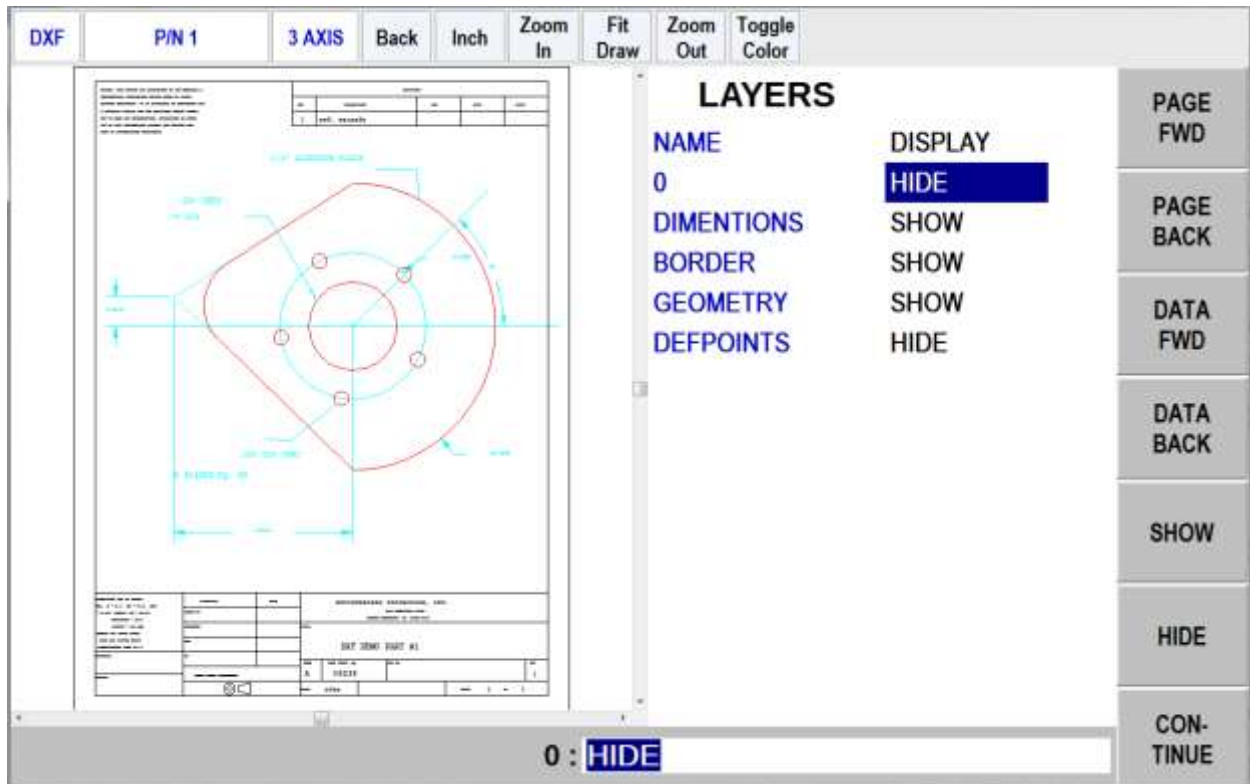


FIGURE 5.2.2 The first screen you see when the DXF Converter launches. Notice that the entire drawing, including borders and dimensions, is shown on this first view.

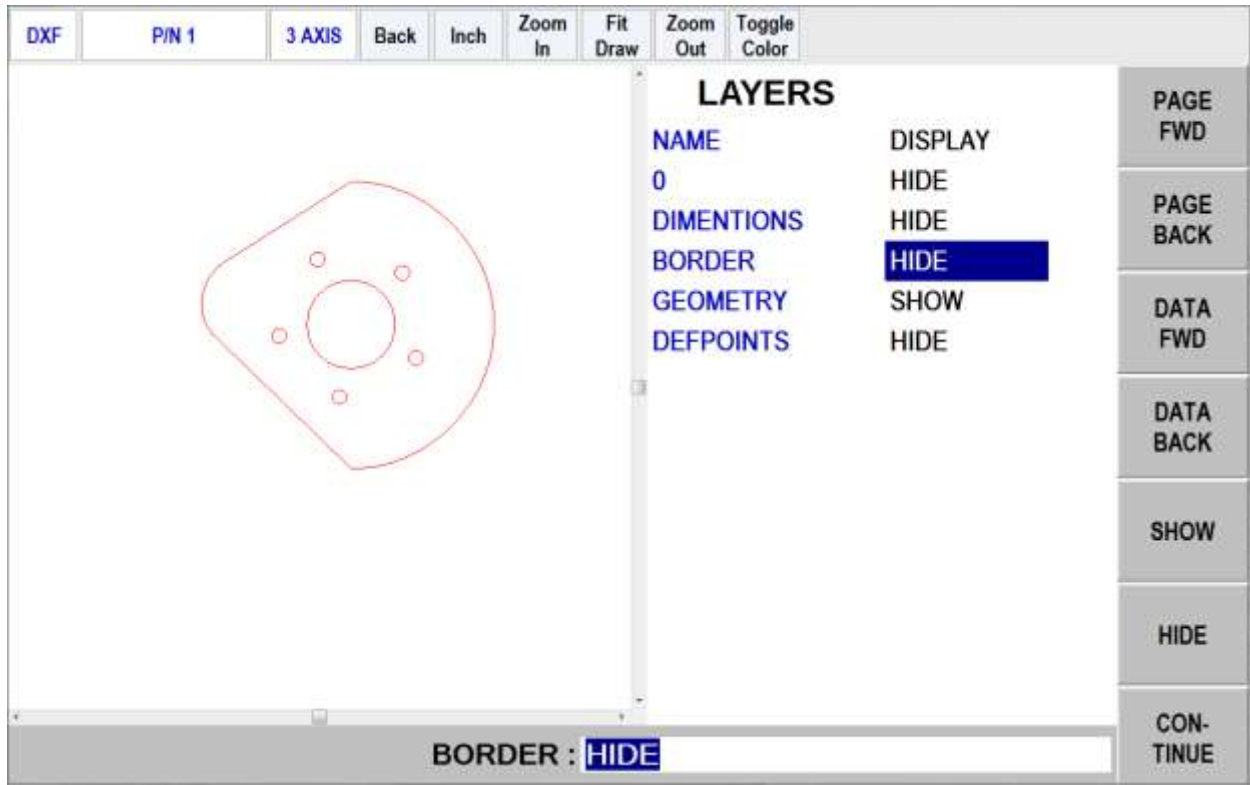


FIGURE 5.2.3 Through a process of elimination, we have isolated the view of what we want to machine.

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:

Toggle Color 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.

See section 5.5.2 for an explanation of all keys on the status line.

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 (or Alt + B on your keyboard). However, any additional data you have added will be lost.

5.2.3 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.

5.2.3.1 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.

5.2.3.2 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:

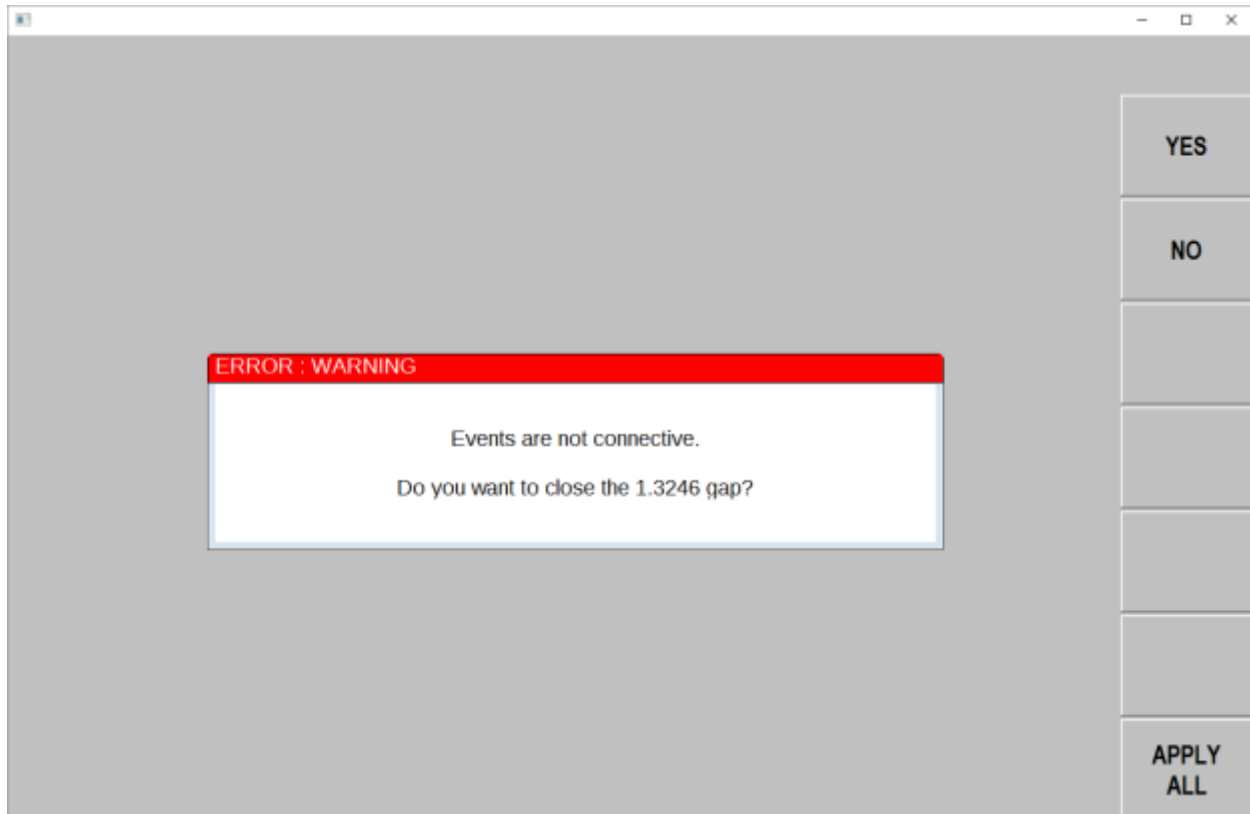


FIGURE 5.2.3.2 A gap is encountered that exceeds the gap tolerance.

- 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.

5.2.3.3 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 5.2.3.2 with the choice of changing the gap tolerance with the Apply All command.

5.3 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.

5.3.1 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.

5.3.2 Choice B: An End Point

This allows you to 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.

5.3.3 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.

5.3.4 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 can be used to return to this screen from any subsequent screens, however, programmed information will be lost.

5.4 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 on or off 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 and end points of a line or arc. Click on one line or arc at a place nearest to the point you 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 you 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.

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. When clicking a circle the diameter will be shown beside the X and Y pointer location position under the part. Click line again to un-highlight. This feature may be useful to determine if you need to break geometry apart to create your part.

Press or click CONTINUE when editing is complete.

Note – if you continue and go to the next screen and begin to program your part and you realize you need to make an edit, you can go back to this screen by pressing the BACK button and edit without losing your program.

5.5 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.

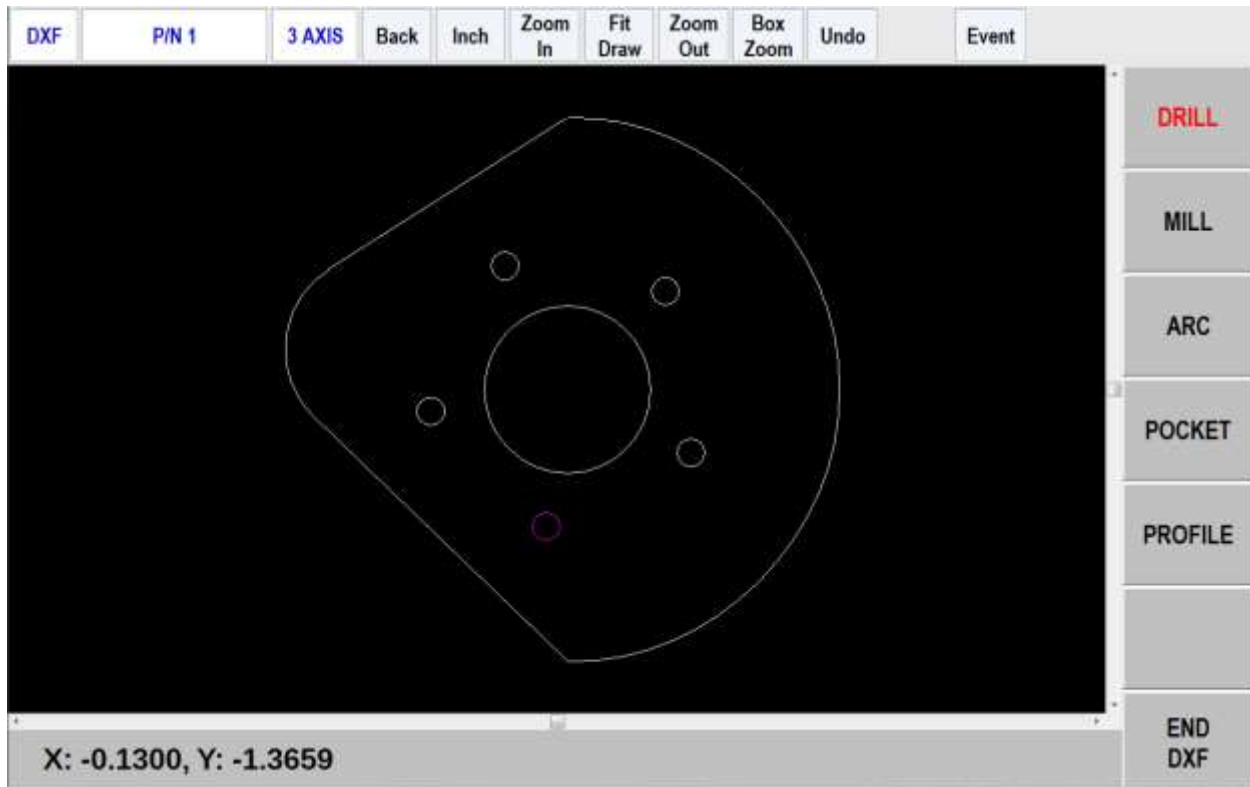


FIGURE 5.5 The Drill event has been selected and now it is a matter of clicking on each circle to load in its geometry.

5.5.1 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. Note – arcs can be used to establish drill coordinates.

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).

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.

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.

5.5.2 Functions in the Status Line

Back. The BACK button allows you to go back to a previous screen.

Inch. The INCH or MM button allows you to switch between inch and metric.

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 Draw. 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 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.

5.5.3 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 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.

5.5.4 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 5.2.3).
- 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.

5.5.5 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 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.

5.5.6 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.

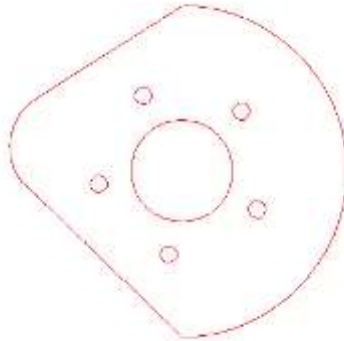
Both ways provide you with the familiar, prompted interface of the ProtoTRAK, but we recommend that you choose the first way – stay in the DXF Converter and click the Event button.

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.

5.6 Sample Exercise

This exercise programs the part shown here. Use the Show/Hide command on the layers of this part program until it looks like this.



Continue and use the center of the circle as the absolute 0 reference point (see section 5.3.3).

Now you can start programming:

1. Select Drill. Note how the Drill softkey text changes to red to signal this button is active. That means the items you click on the screen will be converted into ProtoTRAK Drill events.
2. Click the small circles that make up the bolt hole pattern one by one. Notice how the color changes from white to purple when selected and then to blue when you are done with that feature. 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 in our ProtoTRAK format, which saves the events you just created. You can also change the file name in the Program Mode.

5.7 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 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 2006).

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.

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 Converter process.

Add End Points for Connective Geometry

The 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 5.7.1, 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, use the Break Geometry feature to break the circle into arc segments.

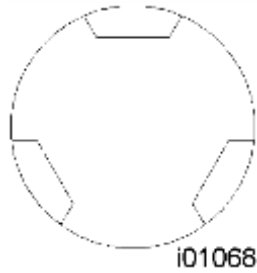
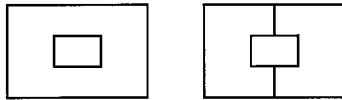


FIGURE 5.7.1 Break the circle into arcs to machine the interior as a Pocket or Profile.

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 5.7.2 below.



i01049

FIGURE 5.7.2 Two simple lines divide the island into two pockets.

6.0 Parasolid Converter

6.1 Introduction to Parasolid Converter

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

With the ProtoTRAK Parasolid converter, you may quickly and easily create ProtoTRAK part programs using the data stored in a Parasolid file.

With the ProtoTRAK Parasolid 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 the part program in the way you direct, leaving you the minimal task of adding the non-dimensional information in the prompted ProtoTRAK format.

6.1.2 Parasolid Extension

Parasolid is a solid model file format owned by Siemens. Most solid graphical CAD systems can be output as a Parasolid file. The standard program or file extension and the one that must be used with the ProtoTRAK is .X_T.

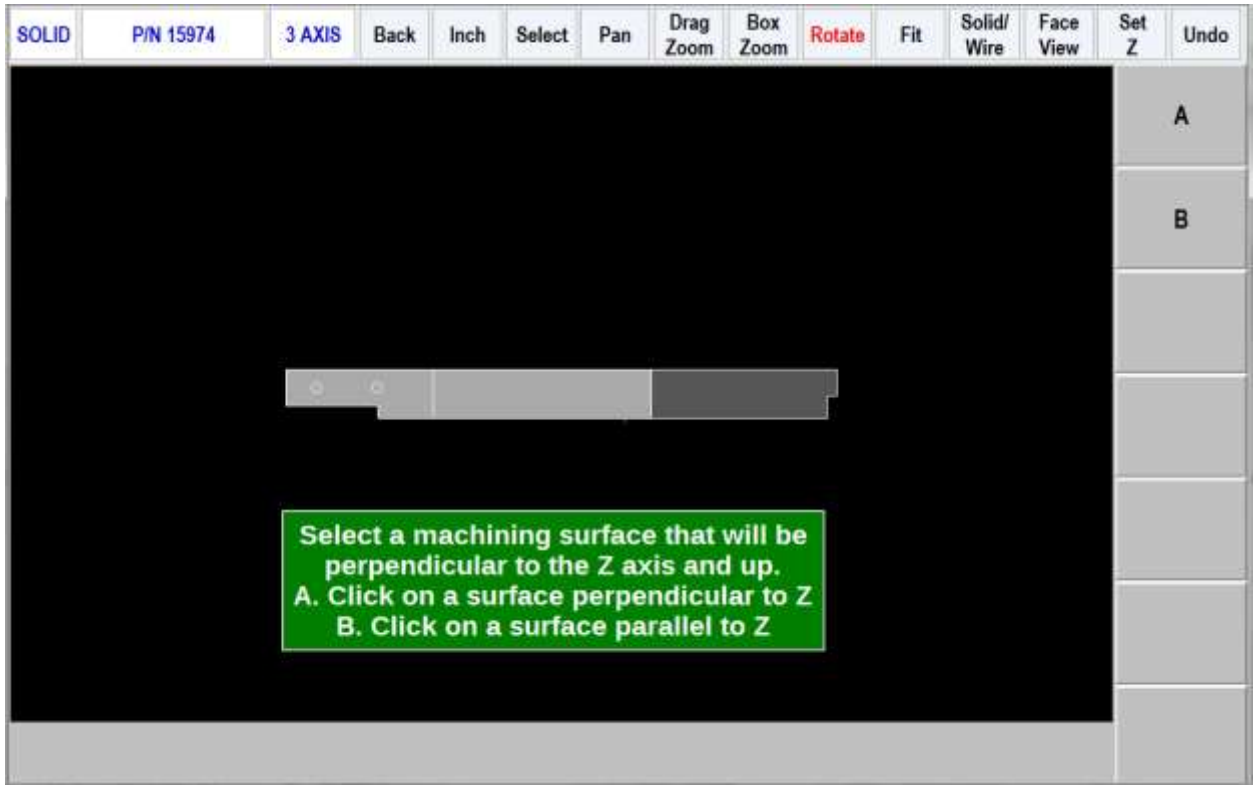
6.2 Creating a ProtoTRAK Program Using the Parasolid Converter

Creating a ProtoTRAK .PT4 program from a .X_T Parasolid is very straightforward. If you are already familiar with the ProtoTRAK DXF converter you will find Parasolid very easy.

6.2.1 Opening a Parasolid Part Program

The Parasolid Converter is launched automatically when you open a Parasolid file with a .X_T extension.

The following screen will be displayed when you open a file.



6.2.2 Status Line

The boxes across the top of the screen change as you move from screen to screen. The list below describes all choices on the status line.

The first line:

SOLID: indicates you are programming with the Parasolid Converter

P/N: shows the opened part number

AXIS: indicates 2 axis or 3 axis programming

BACK: allows user to go back one or more screens

INCH: indicates if dimensions are in inches or mm.

Select: click to enable line, point or surface selecting. Turns off Pan, Zoom and Rotate. You may often be able to select even if this key is not active.

Pan: click to enable panning left, right, up or down. Left click and hold mouse to pan.

Drag Zoom: click to enable drag zoom in and out. Left click and hold mouse to zoom.

Box Zoom: click to enable box zoom. Left click and hold mouse at one location then move pointer diagonally and release to define area to be enlarged.

Rotate: click to enable rotation of part. Left click and hold then move mouse to rotate and release to stop rotation.

Fit: click to resize the part within the display area.

Solid/Wire: Press once to show solid only, again to show wire-frame only and again to show both (the default and generally best display)

Face View: Press to show face view. This will show the view as defined in the Parasolid until you have defined the top view in 3.4.

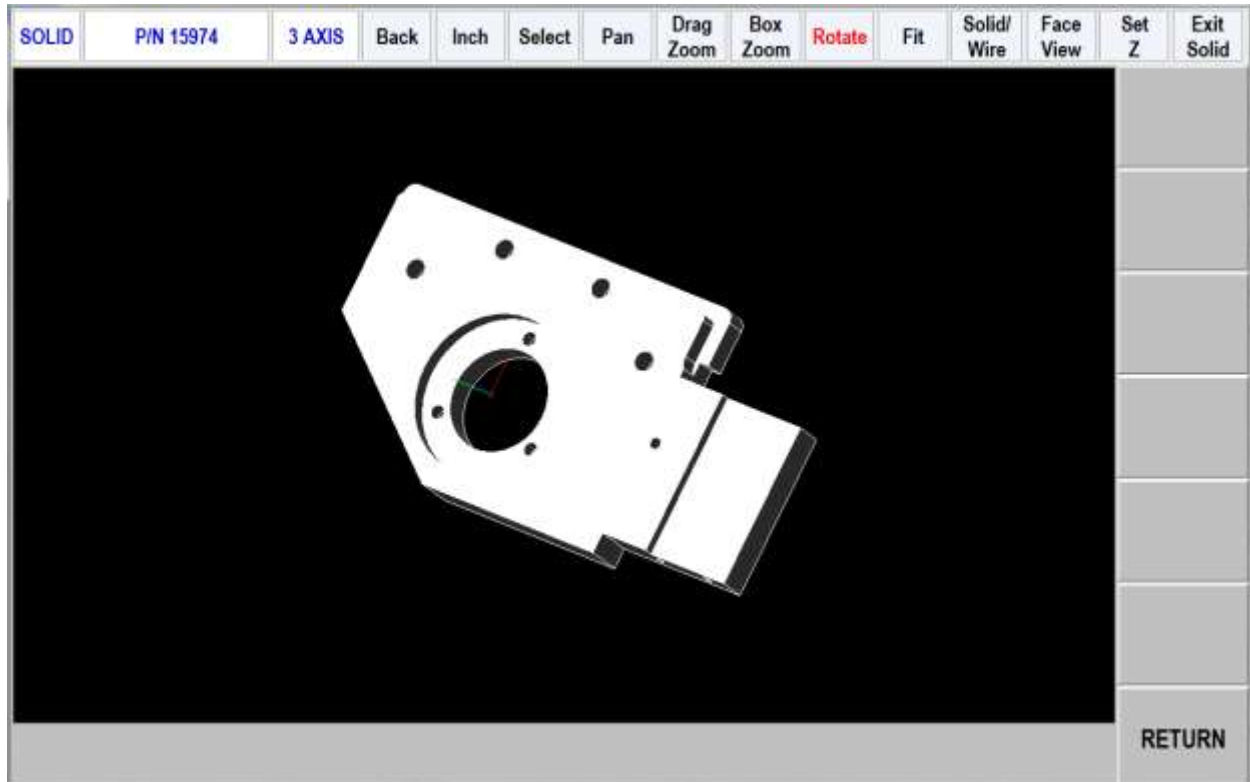
Set Z: defines the Z defaults – see section 6.2.3

Undo: undoes one mouse click

View Solid: allows the user to go back and view the solid 3D model

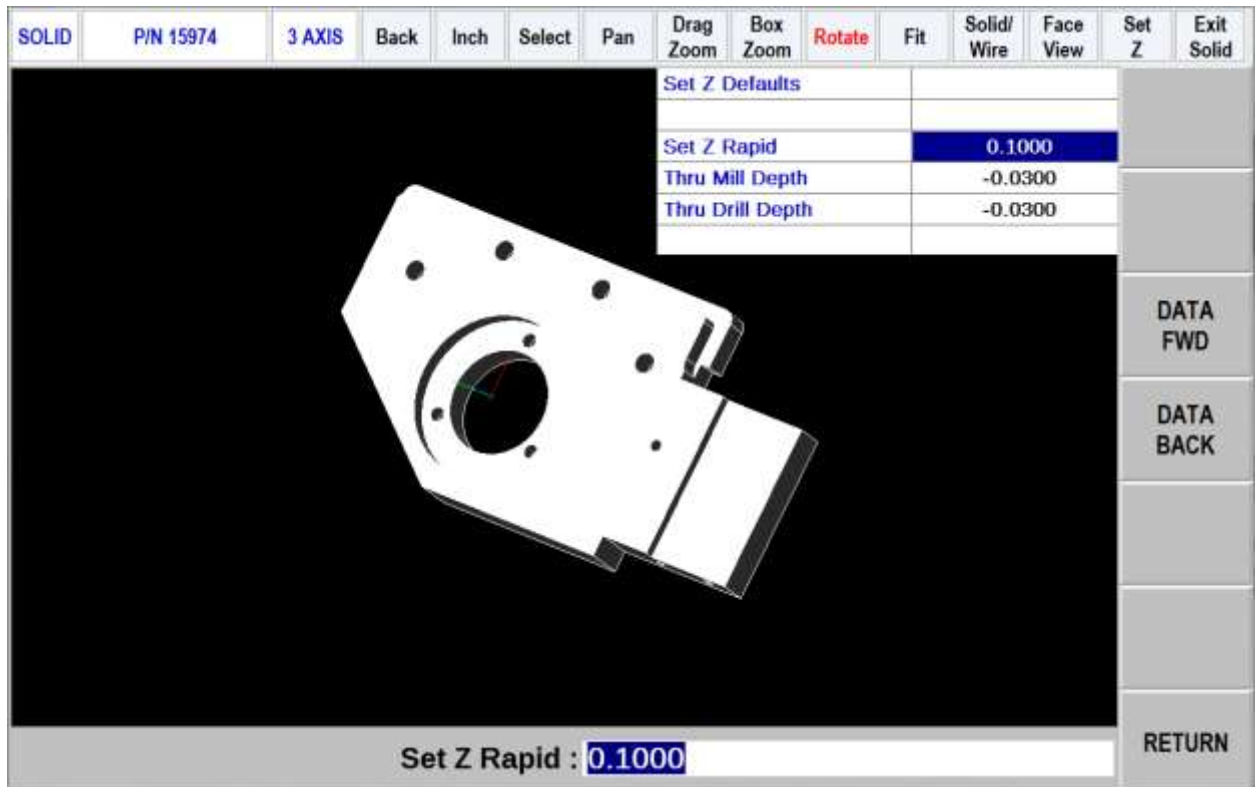
Exit Solid: allows the user to exit the solid model screen and go back to last 2 screens in the software.

EVENT: click to display a drop down box for programming non dimensional information in the events



6.2.3 Set Z Defaults

Click on Set Z to call the following drop down box.



The data within the Parasolid knows the depth of every hole and surface. However, this is often not the depth you want your tool to go to. As you program the part you can choose whether you want the depth of the tool to be the same as the surface you choose or adjusted by the defaults you input in the Set Z drop down box.

Z Rapid: is the rapid to feed transition distance above the surface to be machined

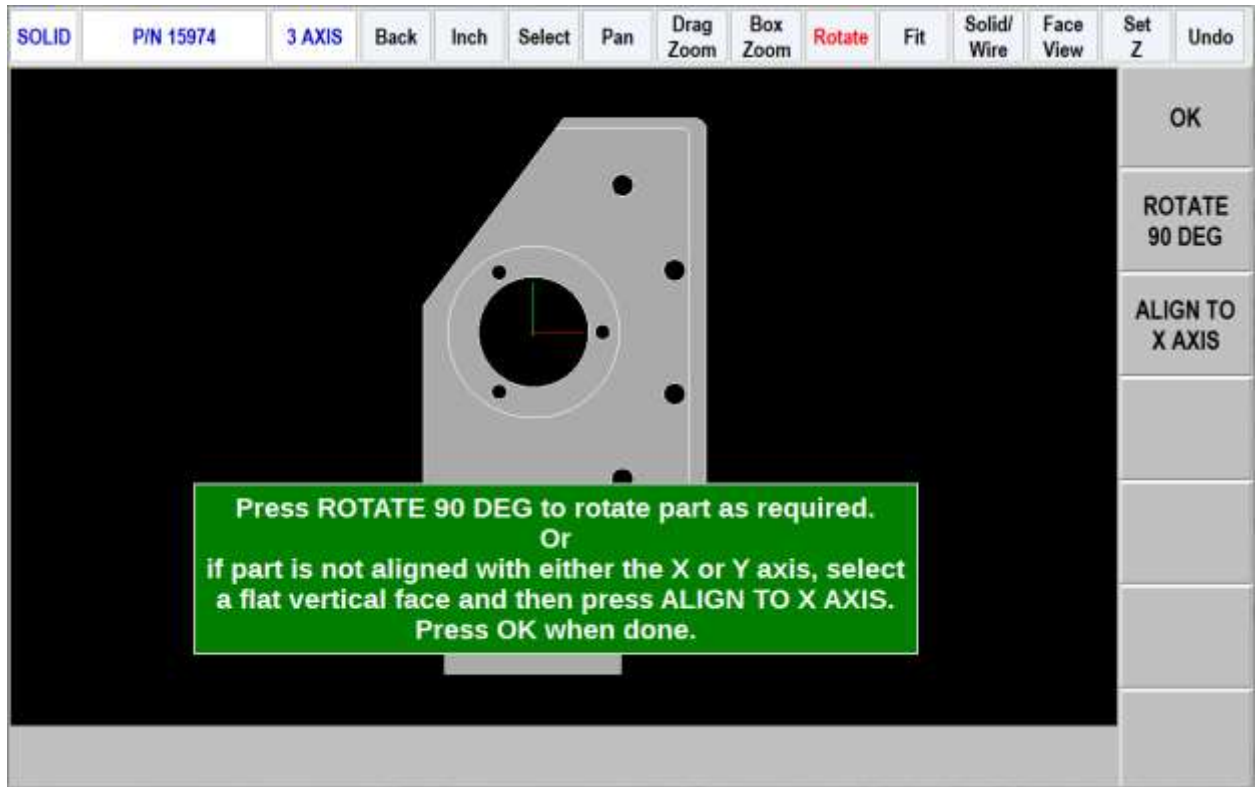
Thru Mill Depth: is the distance (generally deeper and therefore minus) from the final depth surface. This is typical when machining a little below the bottom of a part when cutting around the part perimeter or through the part profile.

Thru Drill Depth: is the distance (generally deeper and therefore minus) from the final depth surface plus the Z dimension of the drill conical point. This is automatically calculated because the Parasolid knows the hole diameter and assumes a 118° point drill.

Click Set Z a second time to remove the drop down box.

6.2.4 Orientating the Part

From the screen in section 6.2.1 the Parasolid Converter is asking you to select which side is up on the part you are programming. Nearly always just select A, rotate the part to identify the up surface, click on that surface and click OK. Occasionally the up surface will be sloped (and therefore not perpendicular to Z) so click on B, rotate the part to find a vertical surface (a side or side of a hole), click on it and click OK.



Next, you have the option to align the part relative to your X and Y axis. If needed, you can click on a flat surface to highlight it, and then click on ALIGN TO X AXIS to line up that straight edge to the X axis. From there, you can rotate the part by clicking on ROTATE 90 DEG. Click OK when satisfied. This is now your face view.

Note that you do not have to use the Rotate or Align features if you are already satisfied with how your part is oriented.

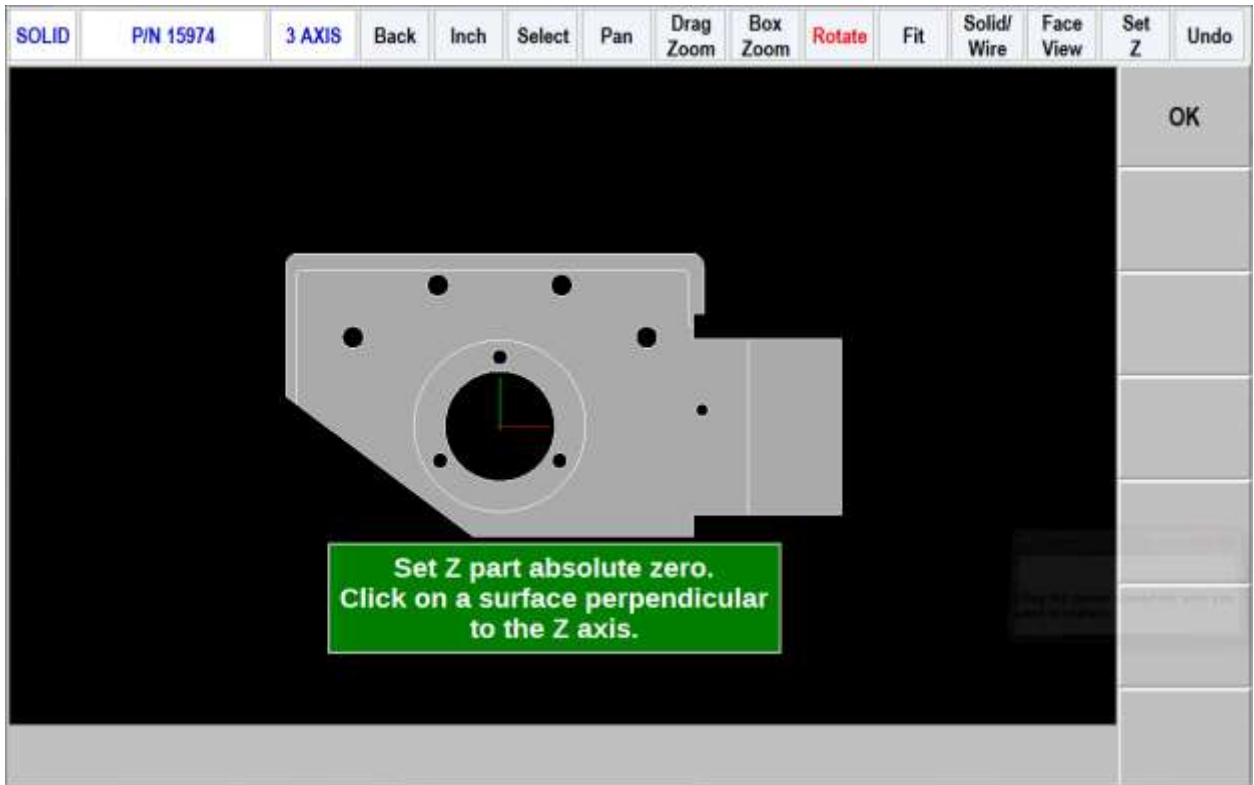
6.3 Setting Program Absolute Zero

6.3.1 Z Absolute Zero

When the part is properly oriented, the Parasolid Converter will ask you to set the part absolute zero. Rotate the part if necessary to show the Z absolute 0 (for example turn it over if the bottom of the part is your zero), click on the surface and click OK.

Notice the system is now in its two dimensional DXF format. This allows for much simpler selecting and programming.

The screen now shows:



Next set your absolute zero for the X and Y axis. The following choices are available. See figure below.

6.3.2 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.

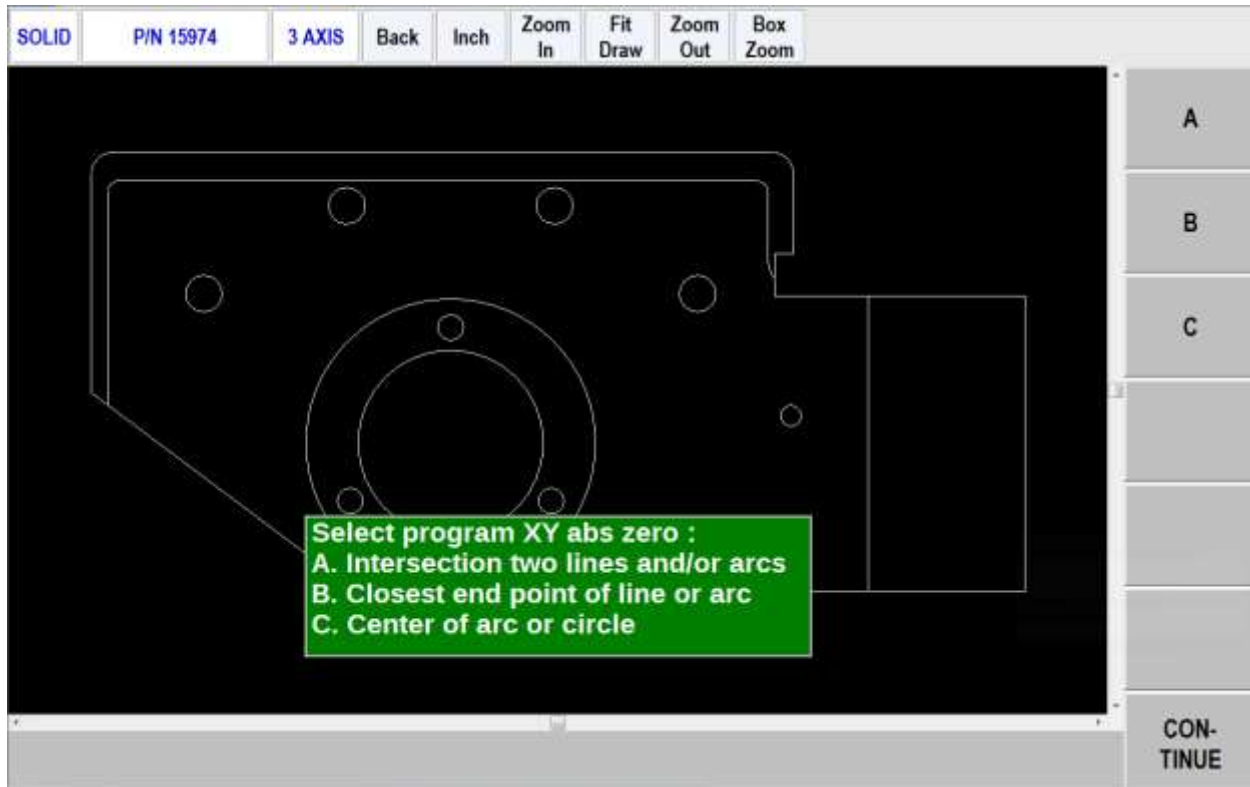
This choice also allows the use of the intersection of two lines that meet off of the given geometry. If this is the case, you will not be able to see the red mark where the new absolute zero reference is set.

6.3.2 Choice B: An End Point

This allows you to 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.

6.3.3 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.



6.4 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 on or off 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 and end points of a line or arc. Click on one line or arc at a place nearest to the point you 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 you 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.

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. When clicking a circle the diameter will be shown beside the X and Y pointer location position under the part. Click line again to un-highlight. This feature may be useful to determine if you need to break geometry apart to create your part.

Press or click CONTINUE when editing is complete.

Note – if you continue and go to the next screen and begin to program your part and you realize you need to make an edit, you can go back to this screen by pressing the BACK button and edit without losing your program.

In the example below, lead on and off moves where added with the New Point feature and show circled in red.

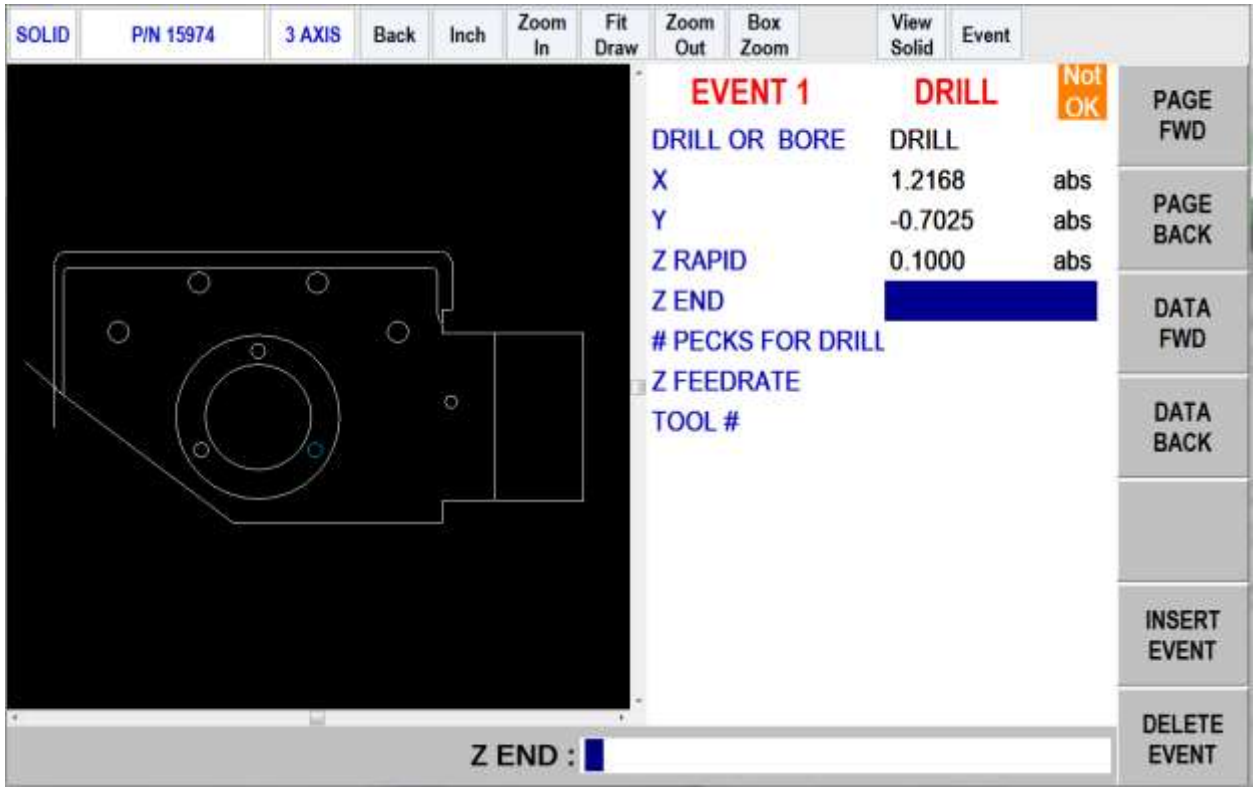


6.5 Converting Geometry into a ProtoTRAK Part Program

Now that the ProtoTRAK Parasolid Converter has the geometry, absolute reference and proper edits for the part, programming is just a matter of selecting the events. With the ProtoTRAK Parasolid Converter, you have total control over how the part is made - just like regular ProtoTRAK programming.

6.5.1 The basic steps are:

1. Pick the event type.
2. Pick the geometry on the drawing (go to the solid if you want a better representation of the part). The X and Y dimensions will be loaded into the event type you choose.
3. Click to show the drop down event. Click on the Z RAPID, click View Solid in the second status line, click on the surface that Z Rapid is measured from and click WITH OFFSET to load the appropriate value for Z RAPID.
4. Click on Z END (if not already highlighted), click View Solid, click on the surface that is the same as Z END and click WITH OFFSET (for through events) or NO OFFSET (for exact depths). See note below for countersinks.



2. Fill in the rest of the prompts to complete the event.
3. Click EVENT to go back to the Select Event screen.
4. Repeat 1 through 5 for all geometry and events.
5. Click END SOLID when complete.

Countersink Note: For Z end on a countersink, zoom in and click the conical sloped surface. The parasolid knows the diameter and slope angle of the surface and will automatically calculate and load the proper Z dimension for the countersink point into the Z END prompt.

Drill Tip Comp –Note the softkey labeled as either DRILL TIP COMP or NO TIP COMP. This can be used to toggle whether the system will compensate for the conical tip of the drill bit (common 118° is assumed).

6.5.2 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).

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.

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.

6.5.3 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 Parasolid 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 counterbore the same five locations. The locations are the same, but you want to use a different tool 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.

6.5.4 Chaining

Chaining is a powerful feature of the Parasolid 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.
- 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.

6.5.5 Color Schemes and Status Flags

Throughout the conversion process on the 2D DXF presentation the 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". The part program will not run unless all events are OK.

6.5.6 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 Converter, click on Event.

2. End the Parasolid Converter and go into Program mode.

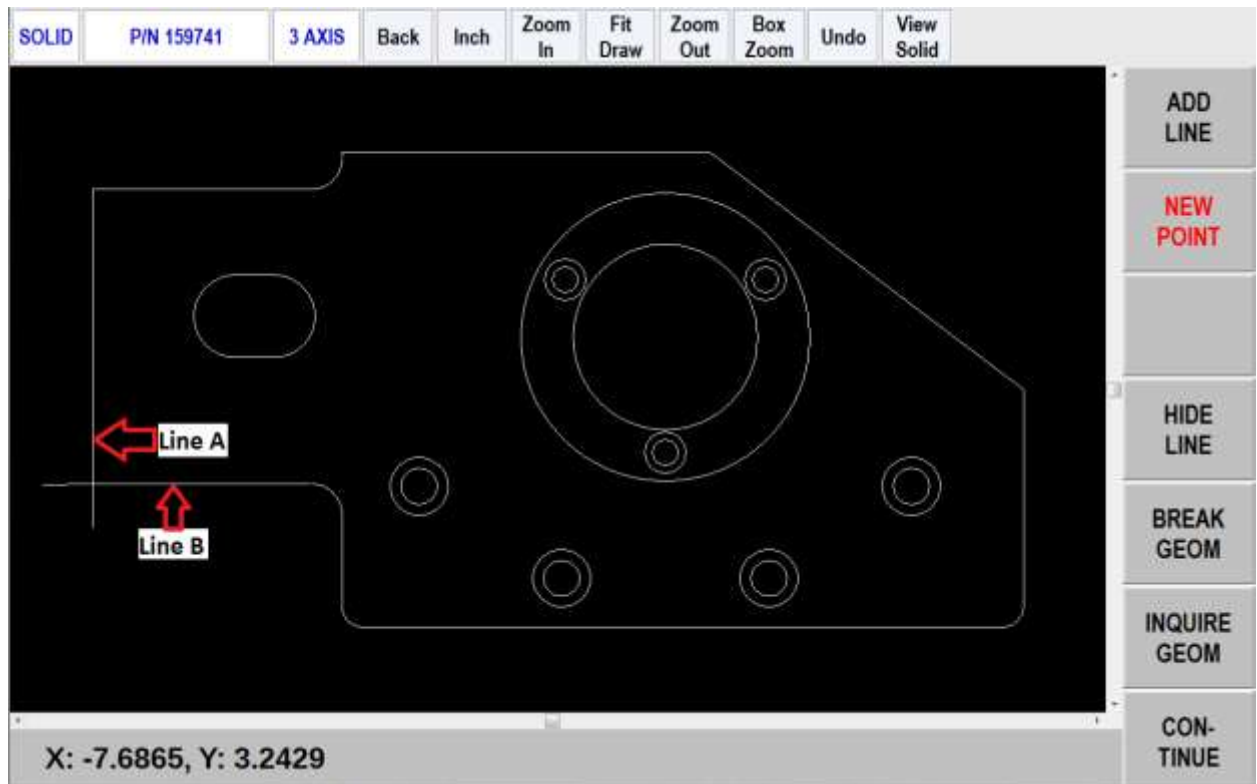
Both ways provide you with the familiar, prompted interface of the ProtoTRAK, but we recommend that you choose the first way – stay in the Parasolid Converter and click the Event button.

We have two reasons for this recommendation. First, we have noticed that when we use the Parasolid 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 Parasolid 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 Parasolid 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 Parasolid 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.

6.6 Sample Exercise

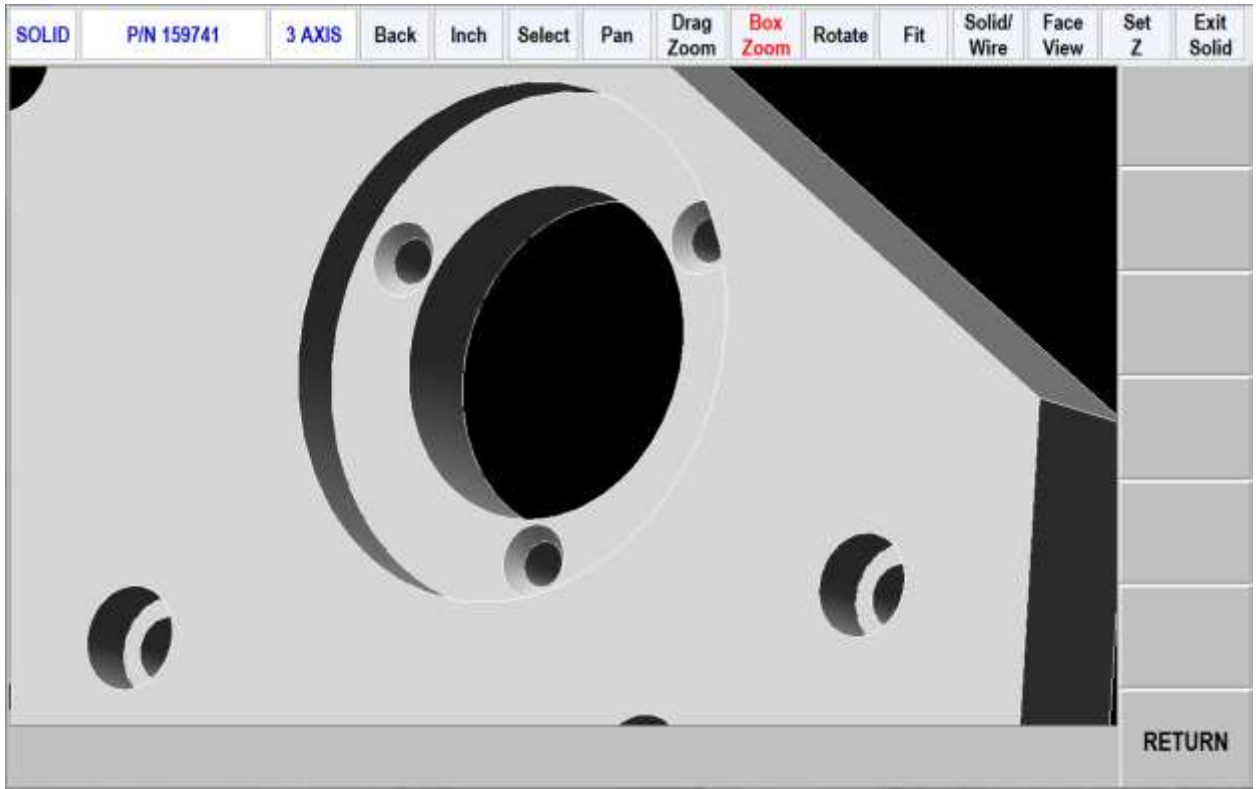
This exercise programs the part shown in this manual using the ProtoTRAK Parasolid Converter.



1. Open part number 159741.X_T
2. Click A.
3. Rotate part to show side with 4 counterbored holes.
4. Click on top surface with the counterbored holes to declare it is up.
5. Click OK.
6. Click TRANS AXES 3 times to orientate part as shown in the figure above.
7. Click OK.

8. Click on the top surface to set Z absolute zero.
9. Click OK. Screen shifts to 2D DXF format.
10. Click C.
11. Click on either large circle line. Notice a red box at center of large circles. This is the X and Y absolute zero.
12. Click CONTINUE.
13. Click NEW POINT.
14. Click near bottom of line marked A, then drag a line down to form a ramp on move for machining the perimeter as shown in the figure above.
15. Click near the left side of line marked B, then drag a line to the left to form a ramp off move for machining the perimeter as shown in the figure above.
16. Click CONTINUE.
17. Click PROFILE to mill around the outer perimeter.
18. Click YES to chain.
19. Click the ramp on line created in step 14, then click the vertical line above it marked A above. The entire profile should now be purple.
20. Click EVENT to activate the drop down event box.
21. Press DATA FWD button to get to the Z RAPID prompt.
22. Click View Solid key in second status line.
23. Click the top surface as the reference from which Z RAPID is measured.
24. Click WITH OFFSET. The display goes back to the event screen and Z rapid is 0.1000 abs (our Set Z default).
25. Click View Solid again.
26. Rotate the part to show the bottom of the part.
27. Click the bottom surface.
28. Click WITH OFFSET because we want to machine a little lower than the bottom.
29. Load in rest of the event information as you would in ProtoTRAK programming. Use tool offset – LEFT and Tool 1.
30. Click EVENT to eliminate the drop down box.
31. Click POCKET.
32. Click YES to chain.
33. Click a line and an adjacent arc in the oval pocket.
34. Click EVENT drop down box.
35. Press DATA FWD button to get to the Z RAPID prompt.
36. Click View Solid, rotate to see the top of the part.
37. Click the top surface and click WITH OFFSET.
38. Click View Solid.
39. Click the bottom surface of the pocket (zoom if you need to) and click NO OFFSET because this is the exact depth that you want.
40. Load in the rest of the event information and use Tool 1.
41. Click EVENT to retract box.
42. Click POCKET, and NO to chain.
43. Click the larger circle around X/Y abs zero.
44. Click EVENT.
45. Data forward to the Z RAPID prompt and just input 0.1000 because you know where the top of the part is.
46. Click View Solid and zoom in a bit.
47. Click the surface that is at the bottom of the larger circle which is the same as top of the 3 countersink holes, then click NO OFFSET.
48. Load in the rest of the event information and use Tool 1. Notice the radius was input automatically because the Parasolid model knows this.

49. Click EVENT to retract the box.
50. Click PROFILE and NO to chain.
51. Click the smaller through hole around X/Y abs zero.
52. Click EVENT.
53. Press DATA FWD button to get to the Z RAPID prompt.
54. Click View Solid, click on the surface at the top of the 3 countersink holes and click WITH OFFSET.
55. Click View Solid, rotate the part to see the bottom, click on the bottom surface and click WITH OFFSET to go a little deeper.
56. Load in the rest of the event information and use Tool 1.
57. Click EVENT to retract the box.
58. Click DRILL.
59. Click the 3 inner circles in a bolt hole pattern around X/Y abs zero that represents the drilled holes.
60. Click EVENT and then input 1, SET for drill.
61. Press DATA FWD button to get to the Z RAPID prompt.
62. Click View Solid, click on the top surface of the holes/countersink and click WITH OFFSET.
63. Click View Solid again, rotate to see the bottom, click on the bottom and click WITH OFFSET to drill through the 0.030" offset plus the drill point length. Notice the total depth of the drill point is -1.1231 in a 1" plate.
64. Load in the rest of the event information and use Tool 2.
65. Click EVENT to retract the box.
66. Click DRILL.
67. Click the 3 outer circles in the bolt hole pattern around X/Y abs zero that represent the countersink holes.
68. Click EVENT and input 1, SET for drill.
69. Press DATA FWD button to get to the Z RAPID prompt and View Solid.
70. Click the top surface of the countersink and click WITH OFFSET.
71. Click View Solid again, click on the sloped conical surface of one of the countersinks, as shown below, and click NO OFFSET. Notice the Z END dimension is where the tip of the countersink must go.



Note the softkey labeled as either DRILL TIP COMP or NO TIP COMP. This can be used to toggle whether the system will compensate for the conical tip of the drill bit (common 118° is assumed). Toggling this will display the difference in depth where the Z DEPTH is displayed.

72. Load in the rest of the event information and use Tool 3.

I'll bet you have a good feel for this by now. Finish up the part by drilling and then counterboring (perhaps with a circle profile) the four holes.

When finished, click END SOLID to create a normal ProtoTRAK.PT4 program that you can run or edit as any other.

7.0 Verify Part

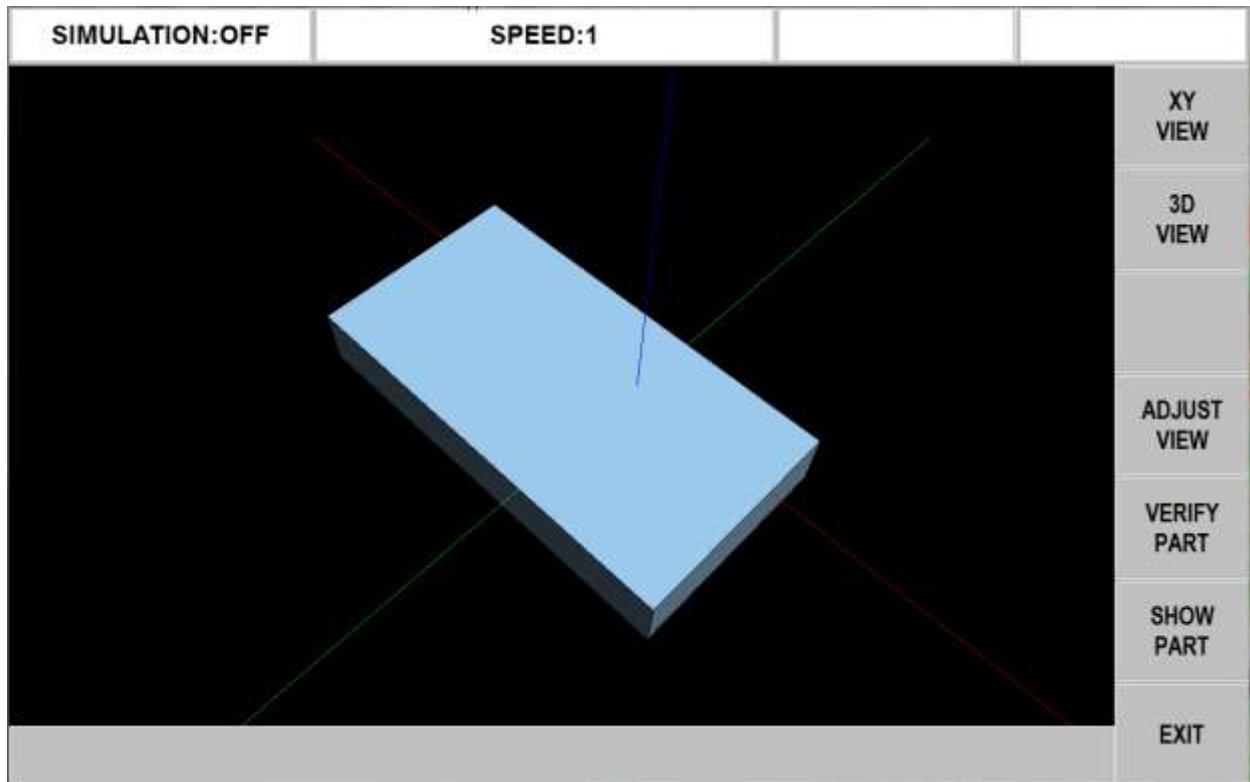
Verify Part is used to see a solid graphical representation of the tools making the part. It should be used as a tool (along with tool path) to verify the part is programmed correctly. Please note that you must define the diameter of your tool along with your type of tool (finish end mill, ball end mill, drill, etc.) you are using when setting up your tools in tool setup. This allows us to draw the correct shape for the tools you are cutting with.

Press VERIFY PART and the following soft keys will show:



Press DEFINE STOCK if you wish to input dimensions of your rectangular stock relative to the programmed absolute zero. We automatically calculate and define the stock slightly larger in the X and Y axis. The Z bottom value we calculate will be based off of the largest negative value we see in the program. This means if you program a drill to go through the part it will make this value the Z bottom value. This can be adjusted by changing the value of the Z bottom dimension. You can manually change the size of your stock by modifying any of the values. If you do modify your stock size and want to get back to the values we calculated or if you modify the program after we calculated the stock the first time and want to update the values then press the AUTO button under define stock. Press RETURN once you define your stock.

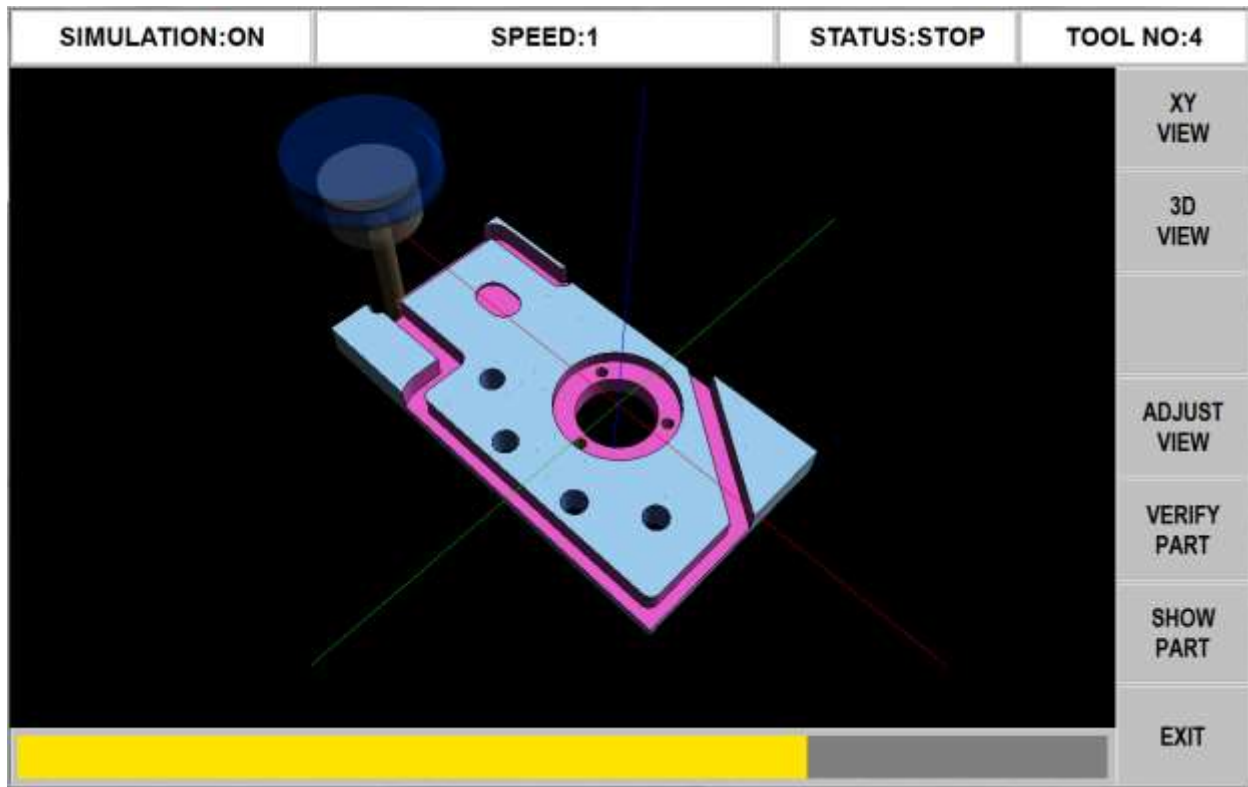
Press MAKE PART and the display shows your stock material.



The status line shows whether simulation is on or off, the speed of the verification, the status of STOP or GO and the tool number for the tool that is being simulated.

The first 5 softkeys are used to adjust the part view and can be pressed before the simulation or during the simulation when the tool is cutting the part. If you are using a mouse you may left click and hold to rotate the stock as it is being cut or once the part is finished. The right button on the mouse can be used to pan the part on the screen and pressing the middle button or scroll wheel can be used to zoom in and out on the part.

Press VERIFY PART to launch the solid graphic part verification routine as shown below. Before or during the verification, press the feed up and down button (use the U or D button when on offline software) to speed up or slow down the machining representation. There are 24 different speeds to choose from. The default speed will work fine for most parts. If the tool moves too fast, press the down feedrate override button one or more times and then press VERIFY PART again. You can adjust the speed on the fly as well. If you press the feed down button 4 times, it will go into single step mode where you can move the tool one move at a time through the verification. Press the GO button each time to move through each step. You may stop verification by pressing the STOP key and continue again using the GO key (use S and G on offline).



Press SHOW PART to simply show the final result.

Press EXIT or MODE to exit the verification and go back to the DEFINE STOCK page.

7.1 2 Axis Verify Software

Input Z End For Verify: Verify is a 3-D solid graphical representation and 3-D, of course, needs Z data as well as X and Y. However, on 2-axis ProtoTRAK KMX programs only X and Y dimensions are programmed. Activating the Input Z End For Verify feature adds a Z END prompt to each event when programming in 2-axis. This input is used only to enable the Verify features.

To activate, go to the Event 0 screen, data forward to the Input Z End For Verify option with the Up and Down Arrow keys and press the YES soft key. The Z END value only needs to be entered on the first event if multiple events are at the same depth. For example, if your program has 10 events with events 1-5 machined at a depth of $\frac{1}{2}$ inch and events 6-10 at $\frac{3}{4}$ inch, you only need to enter a Z END value in events 1 and 6.

TRAK Machine Tools
Southwestern Industries, Inc

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.