

=====  
=  
= Tape-to-Shape V2.0 =  
=  
=====

by

POINT CONTROL CO.

2468 West 11th Avenue  
Eugene, OR 97402  
(503) 344-4470

Printed June, 1988

Copyright 1988 by POINT CONTROL CO

Not to be reproduced in any form without written permission from  
Point Control Co

Point Control Co. assumes no responsibility or liability to the  
licensee or any other person or entity with respect to any damage  
caused or alleged to be caused by the use of Tape-to-Shape.

-----  
Contents  
-----

Introduction	
Overview.....	1
Tape-to-Shape Functionality.....	2
How to use this manual.....	3
Installation	
Overview.....	1
Installing Tape-to-Shape.....	2
Operation	
Preparing to use Tape-to-Shape.....	1
Tape-to-Shape menu selections.....	2
Interpreting a CNC code file.....	3
Viewing the Interpreted CNC program.....	4
Machine Support	
Creating a TTS Interpreter.....	1
What type of code can be supported.....	1
Who should create code Interpreter files.....	2
Steps to create a TTS Interpreter.....	3
Files used by Tape-to-Shape.....	4
The TTS file.....	4
TTS file structure and components.....	4
@Sections.....	5
Sub-Sections.....	5
The TTS file sections.....	6
Geometry, Move sections.....	6
Arc Sub-Sections.....	6
User_Cmdnd sections.....	7
Skip current Block.....	7
Identify Torch or Attachment section.....	7
The TTS file Words.....	8
Variable Words.....	8
Option Words.....	9
Use of Variable and Option Words.....	12
TTS specific Machine File Items.....	16
TTS Interpretation Sequence.....	20
Hints Tips and Tricks.....	21
Common Errors, Troubleshooting.....	22
Example TTS sections.....	23
MILL.TTS.....	24
LATHE.TTS.....	26
PUNCH.TTS.....	27

Applications

- Reviewing CNC programs.....1
  - Enter the CNC program.....1
  - Create a Job Plan file.....1
  - Interpret the CNC program.....2
    - Example #1 (Condensed).....2
    - Example #2 (Raw).....3
  - Viewing the Interpreted programs.....4
- Converting CNC programs.....5
  - Overview.....5
  - General Information.....5
  - Cutter Compensation.....5
  - Canned Cycles.....5
  - Tool Changing.....6
  - Punch Press Conversion.....6
- Creating CAD data.....7
  - General Information.....7
- Using TTS to Teach NC programming.....8
  - Suggested Curriculum.....8

-----  
Introduction  
-----

Tape-to-Shape will "close the loop" by transferring manufacturing data from the shop to the SmartCAM system. The CNC code files you create either with SmartCAM or by some previous means represent a significant investment in creating a database of part geometry and successful manufacturing processes. Tape-to-Shape lets you re-incorporate this data back into the computer aided design and manufacturing process.

Tape-to-Shape will read an existing CNC code file and create a Shape file (.SH2 or .SH3 extension) for use with the SmartCAM system. Tape-to-Shape can be described as a "code interpreter" that processes an existing CNC code file to re-create the SmartCAM Shape source file from which an equivalent CNC code file could be generated.

Tape-to-Shape should not be thought of as a "back plotter." SmartCAM's graphics display and review capabilities are an important part of generating the CNC code file correctly. "Back plotting" is of very limited use when compared to the opportunities made available by Tape-to-Shape's code interpretation ability.

Tape-to-Shape is an ideal way for manufacturers to take advantage of new tooling, different machines, and implement or document engineering changes via a CAM Connection. The increased flexibility Tape-to-Shape provides, allows outdated machines to continue to be cost effective.

You can look at Tape-to-Shape combined with SmartCAM and other available options as the ultimate translator. Once an existing CNC program has been processed by Tape-to-Shape, it can be used in many ways.

Several suggested uses for Tape-to-Shape are:

1. Review or modify existing CNC programs, graphically.
2. Convert existing CNC programs for use on different machines.
3. Transfer an existing "part geometry database" from a CNC program to a CAD system using a CAM Connection.
4. Tape-to-Shape also provides educational institutions the ability to fully utilize the equipment they have purchased or had donated to them. As a teaching and training tool Tape-to-Shape allows students to practice manual CNC programming for any given industrial machine and check results graphically. They can then re-code the resulting Shape file to create a program to run on an available CNC training machine.

-----  
Tape-to-Shape Functionality  
-----

Tape-to-Shape can interpret geometry from CNC files for a wide range of machines that use an explicit command structure such as G and M codes. The CNC code file to be interpreted by Tape-to-Shape must exist as an MS-DOS text file. SmartCAM's Communicate - Read\_Tape option can be used along with an interfaced tape reader to read the CNC file from an existing tape to the text file. Before you process a tape, you must also know what machine the tape is for and have a properly configured SmartCAM "Code Generator" along with the necessary TTS Interpreter file. You will also need a Job Plan file that describes the tooling to be used. It is possible to use one of the supplied "generic" Job Plans for a quick review with the condense mode off.

The completeness of the resulting Shape file created by Tape-to-Shape is dependant on the content of the CNC code file processed, for example some machines may support cutter diameter compensation commands. When these commands are included in the CNC code file, Tape-to-Shape can re-create the original part profile geometry for the path programmed for tool centerline. If the compensation commands are not included you will need to use the Shape module's Tool\_Path - Wall\_Offset function to re-create the part profile from the tool's centerline path.

Note that the results of a Tape-to-Shape transfer should always be reviewed in SmartCAM's Shape module before you use this Shape source file for other purposes.

NOTE: Custom TTS Interpreters are included as part of a Custom Code Generator and are available for SmartCAM and Tape-to-Shape. The price and ordering process are the same as for any Custom Code Generator from Point Control Co.

-----  
HOW TO USE THIS MANUAL  
-----

This manual provides you with the information you need to correctly operate your Tape-to-Shape software.

The table of contents outlines the entire manual for you. The manual has been divided into five chapters. Note that the sections have been paginated individually for ease of updating or adding information. Should you receive an update, please insert it in the appropriate section in this manual.

To properly make use of Tape-to-Shape, read this manual completely. If you are not yet comfortable with the operation of SmartCAM, read the SmartCAM documentation and continue to explore the system until you are confident of your abilities. Pay particular attention to SmartCAM's file structure and paths.

If you are already a confident SmartCAM user, proceed to read this manual and install the Tape-to-Shape software on your system.

When using this manual you will find the following typographical conventions:

"Screen prompts appear in bold within quotation marks."

Instructions to you are underlined.

Keyboard entries you are to type in appear in boldface.

"CNC program" refers to any Numerical Control media format.

We at Point Control Co believe that Tape-to-Shape will increase both your productivity and your profit margin. Inform us if you encounter a problem, and let us know about interesting applications you may develop for our products. Good luck.

-----  
Installation  
-----

This chapter will guide you through installing Tape-to-Shape into your SmartCAM CNC programming system.

Once installed, Tape-to-Shape will operate from SmartCAM's Main Menu. Menu items in Tape-to-Shape can be selected in the same manner as in SmartCAM: Type the first letter of the function; or pick the function with a mouse; or scroll Up and Down using the arrow keys to highlight the function you want and press Enter.

It is not possible to operate Tape-to-Shape alone, it must be used in conjunction with SmartCAM. Therefore the hardware requirements are the same. You must have a 100% IBM compatible computer with at least 640K, a hard disk and a math co-processor such as an 80287.

Follow the instructions in the Installation chapter of the SmartCAM Reference Manual, to install and configure SmartCAM version 4.0x on your system. Test SmartCAM for correct operation, then follow the instructions below for installing Tape-to-Shape.

NOTE: We encourage you to use and become familiar with SmartCAM before you attempt to use Tape-to-Shape.

SmartCAM Version 4.0x is installed in a \SM4 directory and three sub-directories labeled DEV, SMF and DATA.

The \SM4 directory contains setup, batch, executable, and miscellaneous files necessary to operate the system.

The DEV subdirectory contains the device drivers needed to communicate to the various hardware items.

The SMF subdirectory contains all the machine and template files needed to generate code for specific machines.

The DATA subdirectory contains Job Plan (.JSF) and Shape (.SH2 or .SH3) files. Sample files are included with SmartCAM and reside in the data sub-directory.

It is possible for you to create more subdirectories to help organize files by customer, machine type or application, etc. To do this you will need to create a new subdirectory using DOS. Use this new subdirectory to save the Job Plan and Shape information.

-----  
INSTALLING Tape-to-Shape  
-----

If you do not have SmartCAM Version 4.0x installed on your system, you should install it before attempting to install Tape-to-Shape. If you have any questions regarding SmartCAM's installation routine, refer to your SmartCAM Reference Manual.

CAUTION: This installation procedure will write over all files that have the same filenames as those contained on the disk. This is especially important in the case of the SMF directory and the SMF files currently on your system. Place the Tape-to-Shape disk in drive A and type `Dir A:SMF\*.SMF`, a listing of all SMF files contained on the disk will be displayed on your screen. If you have any edited .SMF files on your system, that you have saved under the original filename, you should change the name of these files so they do not get overwritten.

NOTE: The following commands must be executed from the DOS prompt with `\SM4` as the current directory.

1. Insert the Tape-to-Shape disk in Drive A.
2. At the DOS prompt `C>`, type `a:install` and press Enter

The computer will copy the files from the Tape-to-Shape disk to the appropriate subdirectories. TTS files are copied to the SMF subdirectory, and all others are copied to the `\SM4` directory.

When the files have been copied, you will see the DOS prompt.

The copied files now reside in their respective subdirectories within the `\SM4` directory. For more information on SmartCAM file structure, see the Installation chapter, page 40, of your SmartCAM Reference Manual.

-----  
Operation  
-----

This chapter will guide you through using Tape-to-Shape one menu item at a time. You will find a brief explanation of each menu selection, its function, and the screen prompts associated with the menu item.

#### Preparing to Use Tape-to-Shape

Before Interpreting a CNC program, you must first create a Job Plan that Shape uses when it displays the toolpath geometry. Tape-to-Shape requires a Job Plan to provide tool information about the CNC program when you use the condensed mode. If you have not already done so, select Job Plan from SmartCAM's Main Menu and create a Job Plan with the appropriate tools for the CNC program you are going to Interpret. More information on setting up a Job Plan is available in the Job Plan chapter of your SmartCAM Reference Manual.

If you are not sure about the CNC program and the tools you need, we have provided generic Job Plan files named TTSMILL.JSF, TTSLATHE.JSF, TTSPUNCH.JSF, TTSEDM.JSF, TTSBURN.JSF, TTSROUT.JSF, and TTSLASER.JSF. Use the appropriate Job Plan file to preview the CNC program.

Previewing CNC programs should be done with the condensed mode set to NO, as this will preserve all the information interpreted from the CNC program. Previewing a program using a generic Job Plan file, will allow you to see the programmed toolpath geometry, even though the tools that are displayed are not the correct tools for the operations being performed. All elements will be assigned to Tools #1 - #32. This will help you identify tool changes because of the color differences and the size of the displayed tools. If your CNC program contains more than 32 tools, all tools beyond #32 will be assigned to Tool #1. After previewing the program, you should create a new Job Plan file with the correct tools and Interpret the CNC program again using the new Job Plan file.

The CNC code file to be interpreted by Tape-to-Shape must exist as an MS-DOS text file. The following section assumes that if you started with a punched tape, you have read the CNC program into SmartCAM using Utilities - Communicate - Read\_Tape. If you need more information on communicating with SmartCAM, see the SmartCAM Reference manual, Utilities page 3. If you have not read in a CNC program we have provided a file for you to experiment with. The name of the files are TTSLRN and TTSLRN3D.

-----  
Tape-to-Shape's Menu Selections  
-----

To learn the basic operation of Tape-to-Shape follow the underlined steps on the following page. If you would like to use the file we have provided (TTSLRN or TTSLRN3D), enter the information at the end of each instruction on the following page.

Select Tape-to-Shape from SmartCAM's Main Menu.

The single-line menu will display four options:

Interpret            Set\_Mode            Dir\_List            Leave

Interpret reads and interprets CNC programs for use within the SmartCAM Shape module.

Set\_Mode will change the default setting for the Tape-to-Shape Interpretation.

After selecting Set\_Mode you will see the prompt.

"Condense will create part profile from tool center and remove extra elements. Condense shape file? Y/N"

The normal default is Yes, which will condense the Shape file and eliminate all unnecessary elements. When the CNC code includes the necessary compensation commands Condense mode will recreate the geometry for the original part profile. Entering No will cause Tape-to-Shape to preserve all the information Interpreted from in the CNC program and include it in the Shape file. The No setting is the mode you want to use to preview CNC programs so that you can see everything that was interpreted from the program.

Dir\_List lets you list the filenames in a specified directory. This is useful for checking Job Plan, Shape and CNC program filenames.

Leave returns you to SmartCAM's Main Menu.

-----  
Interpreting a CNC code file  
-----

1. Select Interpret from Tape-to-Shape's Main Menu.

The screen now indicates whether that you will be Interpreting in the Condensed mode (YES) or the Raw mode (NO).

Also displayed are the next four prompts for the filenames you will enter. As you enter each filename, the input cursor will advance to the next input.

The first prompt asks you to "Enter CNC code file:" Type the filename of the CNC code and press Enter. Example: TTSLRN

The next prompt asks you to "Enter Job plan file:" type the filename of the Job Plan and press Enter. Example: TTSLRN

The third prompt asks you to "Enter Machine define file:" type the Machine Define filename and press Enter. Example: MILL

The final prompt asks you to "Enter Destination Shape file:" type the filename for the shape file and press Enter. Example: TOSHAPE

Note that the destination shape filename is displayed with an extension of sh2 or .sh3 depending on whether the Job\_Plan used to translate the file is a 2D or 3D job plan.

2. Tape-to-Shape now goes to work Interpreting using the files that you specified. While Tape-to-Shape is working, you will see the following message displayed on your screen:

```
" Working...+++++++°°°°°°°°°°°°°°°°°°°°+++++++°°°°°° " 
```

NOTE: An Error report for Arc start and end radii will be displayed if there is a discrepancy between the start and the end of the Arc. This error report will include a copy of the CNC code block, the element #, and the amount of the discrepancy.

3. After Tape-to-Shape has finished Working, you will automatically be returned to the Tape-to-Shape Main Menu
4. Type L to Leave Tape-to-Shape and return to SmartCAM's Main Menu

-----  
Viewing the Interpreted CNC program  
-----

The next step is to go into the Shape module and view the part that was Interpreted from your CNC program file.

Enter the Shape module and read the file that you have Interpreted.

Note that if you used the Raw (uncondensed) Interpretation mode, the shape geometry displayed on your screen is for the programmed toolpath centerline. All tool offsets have been changed to None. Also displayed are User commands that are associated with Speed or Feed changes. When you are using the condensed mode a user command will appear only if the Speed or Feed differs from the Job Plan.

Use the Shape module to view the Shape file created from the Interpreted CNC program and make any needed changes to the part geometry. Usually the first step is to observe the tool path using the View - Show\_Path functions to check for such things as machining sequence, work holding interference and completeness of the machined part.

A step by step view of the tool path is possible using Shape's, Toolpath - Follow functions to view the operations and user commands one at a time.

Use the F7 key to look at the Interpreted CNC program code; use the Pg.Up and Pg.Dn keys to page through the data.

If you would like to print a hardcopy of the Interpreted CNC program data, use the View - Output - List\_Data - Entire - Accept functions to Send the information to your printer.

-----  
Creating a custom TTS Interpreter  
-----

Tape-to-Shape can interpret CNC programs for most CNC machine tools supported by SmartCAM. Tape-to-Shape can be configured by the user to work with a given machine in much the same way that a custom "Code Generator" can be created to generate code from SmartCAM. In fact, one of the two files used by Tape-to-Shape is also used to generate code. This section of the Tape-to-Shape Manual describes how you can modify existing interpreter files to more closely match your programming style or create a new set of interpreter files for a different machine.

What type of code can be supported?

Tape-to-Shape can interpret geometry from a wide range of machines that use an explicit command structure such as G and M codes. An example of some formats that are not currently supported are "tab sequential" and "Bandit II". In these formats the current type of move is implied by the contents of the block rather than given by an explicit command. Note, however, that the SmartCAM system has no problem generating code for these other formats.

Tape-to-Shape can also interpret non-geometry commands such as speeds, feeds, or other miscellaneous commands. These are represented in the Shape file as User\_Cmds elements. Tape-to-Shape also has an option to create a SmartCAM User\_Cmd element for "custom" machine cycles and even sub-routines calls. These will be displayed as User\_Cmds in the Shape module, and except for the "system User\_Cmds," such as @BHC, these custom User\_Cmds will not show the toolpath graphically. If corresponding User\_Cmd sections are incorporated in the template file used to generate code, the contents of the original commands can be re-coded with no loss of meaning to the machine.

Some machine controls provide advanced higher level language features such as variables, expressions, and conditional go-to statements. There is currently no provision for any of these higher level language features or for any special cycles that repeat geometry at some other orientation such as rotated, mirrored, or scaled. Tape-to-Shape can still be applied to CNC programs using these types of commands, but the resulting Shape file will need to be modified Manually to fill in the un-interpreted information.

Any geometry to be interpreted must be explicitly stated using one of the following positioning conventions, absolute, relative, or polar coordinates. Also note that some machines may lack the commands necessary to completely define the original Shape source file.

-----  
Who should create code Interpreter files?  
-----

Before attempting to implement a custom TTS interpreter the user should be thoroughly familiar with the section in the Utilities chapter of the SmartCAM Manual on Creating a Custom "Code Generator"

The creation of a TTS interpreter requires a commitment to work through several cycles of modifying and testing for accurate results. If you can not justify the time or commitment to complete this process, buy a machine specific TTS interpreter through your dealer.

Note that since the release of Tape-to-Shape any custom "Code Generators" purchased from Point Control Co. will include the TTS files necessary for use with Tape-to-Shape.

-----  
Steps to Create a TTS Interpreter  
-----

Usually you will want to start by first creating a "Code Generator" for a given machine before attempting to create the TTS interpreter for that machine. This is because one of the two files used for each process is the same and you will also have a clear understanding of what codes need to be supported.

The two types of files used for a TTS interpreter are (1) Machine files and (2) TTS files. Note that the first file is the same Machine file used as part of a "Code Generator." Sample TTS files are included with Tape-to-Shape to match many of the sample code generators supplied as part of your SmartCAM system.

You will use the Machine Define function included with SmartCAM to modify a machine files (.SMF extension). Note that several new questions have been added to support items specific to Tape-to-Shape.

The TTS files (.TTS extension) can be edited in SmartCAM's Edit\_Plus module. Note that when you installed Tape-to-Shape the sample TTS files were copied to the SMF subdirectory.

Creating a TTS interpreter for a specific machine will usually involve the following steps:

1. Start with a "Code Generator" for the specific machine to be supported by Tape-to-Shape.
2. The TTS file may be created by:
  - a. modifying a similar TTS file from another machine.
  - b. modifying the template file created to support this machine.
3. Use Machine Define to change the items specific to Tape-to-Shape as necessary.
4. Use Tape-to-Shape to process CNC files created by SmartCAM using the related code generator. Compare the resulting shape file created by Tape-to-Shape with the original shape file. Be sure and test all of the cycle you wish to support.
5. Repeat steps 2 - 4 until the shape file created by Tape-to-Shape is functionally equivalent to the original file. Note that they may not be exactly identical but should result in the same toolpath.
6. Once satisfied with step 5 you are ready to test CNC code files not created by SmartCAM. Further adjustments may need to be made to allow for the variations inherent in Manually written CNC programs.

-----  
Files used by Tape-to-Shape  
-----

A Job\_Plan file, Machine file, and TTS file are all required in addition to the CNC code file to be interpreted.

#### The TTS file

Tape-to-Shape uses both the TTS file and the Machine file (.SMF) to interpret which machine codes should result in a particular Shape element. Both of these files must be located on the same disk and directory. The particular TTS file to be used is specified by Item #7 of the Machine file.

The TTS file serves a function in the Tape-to-Shape process similar to the template file's function during code generation. It allows the user to specify how the specific machine commands are to be interpreted and even allows for several optional uses of each command or different commands with similar results. You can think of the TTS file as a "roadmap" that tells Tape-to-Shape where to look for the information that describes each machine move or action.

#### TTS File Structure and Components

Now is a good time to look at a sample TTS file. Look at the sample file at the end of this chapter, or print out one of the sample files included with your Tape-to-Shape package by using Edit Plus.

There are two basic components of the TTS file, sections and words. Each "section" defines the Shape element or action which should result from the associated CNC command. Most sections will also contain the "words" used to define the geometry or action given by the command for that section. Some words will also control "options" for different ways of defining the geometry.

## @SECTIONS

Each TTS file consists of sections that define how each CNC command should be interpreted. Each section is marked by the special character @ followed by the name of the section as described later. Associated with most sections will be the CNC command that when found in the current CNC code block, will activate this section. This CNC command will appear in parentheses following the section name. On separate lines following the section name will be a list of words or commands. Each variable word or command specified in the section is then searched for in the current block and the values for each word in the block are used to define the selected geometry or operation.

Many machines do not require an active move command to be reprogrammed if it has not changed. Such commands are "modal." If the command for a section is considered to be modal, the same section will continue to be used for the following CNC code blocks until a command that changes the section is found.

### Sub-Sections

Some sections may use additional sub-sections which look like the main section labels, but are actually part of the complete definition of a higher level section.

-----  
The TTS File Sections  
-----

A short list of the available sections is included here with more complete explanations and examples given later in this chapter.

Geometry, Move Sections

@RAP() Rapid move.....creates a POINT element  
@POS() Set current position.....creates no element  
@LINE() Line cut.....creates a LINE  
@FIXED() Perform a hole operation..a POINT element  
@ARC-CW() Clockwise arc.....ARC element, (dir=CW)  
@ARC-CC() Counter-clockwise arc.....ARC, (dir=CCW)  
@THREAD() Thread operation.....THREAD element (Lathes)  
@TURN() Turn operation.....LINE element (Lathes)  
@FACE() Face operation.....LINE element (Lathes)

All of the geometry move sections listed above will specify the CNC command inside the parenthesis following the section name.

Example: @RAP(G00)

If there is a second command that is equivalent to the first in the way it should be interpreted, the '=' symbol can be used to make the two sections equivalent.

Example: @FIXED(G82=G81)

Arc sub-Sections

@XZPLANE Define interpretation of Arc Command when XZ Plane active.  
@YZPLANE when YZ Plane active.  
@XYPLANE when XY Plane active.

Note that @XYPLANE is the default and it's use is optional.

These sub-sections are applicable for arc commands when interpreting a file for use with SmartCAM's 3-D Machining module. They are used to specify how an Arc should be interpreted for each of the orthogonal machine planes.

Machine Define items #271 thru 276 are used to identify the commands used to select the active orthogonal plane and to determine the correct interpretation of Arc direction.

User\_Cmd Sections

@USRV User\_Cmd set variable.....User\_Cmd element  
@CUSTOM Custom User\_Cmd list.....Custom User\_Cmd element

Skip Current Block

@SKIP Skip list.....ignore block with one of these commands in it

Identify Torch or Attachment Section

@TORCH() Identify tool in Job Plan..Doff equivalent  
@ATTCHMT() same as @TORCH

The parenthesis following the section name will include the CNC code or command that activates the location offset for the attachment. The line immediately following the section name will be an assignment of the number entered in the Job Plan Loff cell that identifies the attachment or torch tool to be used.

Example.

@ATTCHMT (E02)  
#EVAL(#LOFF=2)

NOTE: See Machine Define item #411 for an alternative method of identifying a torch attachment.

-----  
The TTS File Words  
-----

#WORDS

TTS words fall into three basic categories: (1) they can act as variables to represent the numeric entries required to describe a given machine move, (2) they can specify an option for how the move may be defined, and (3) they can perform an interpreting function. Words are always specified on separate lines of the TTS file following the section name.

A Variable word will always have an associated machine code letter or command. This is given at the start of the line followed by the variable word as follows:

X#XPOS

Where X is the machine code letter or command and #XPOS is the variable word.

This tells Tape-to-Shape to find the matching letter or command in the current block and assign the following numeric value to this variable word. After all variable words in a section have been searched for and assigned, the geometry specified by that section is created using all of the "input values" taken from the CNC code.

#XPOS is an example of a variable word. This variable is used to specify the X axis position for a machine move. Depending on the section and the application you may use any SmartCAM "template word" as described in the SmartCAM Reference Manual Utilities chapter.

A few typical variable words:

#XPOS, #YPOS, #ZPOS  
#XCTR, #YCTR, #ZCTR  
#LNANG  
#LNLEN  
#ARAD  
#STANG  
#TANG  
#SPEED  
#FEED

Option words are used to specify how the geometry for a given section is to be specified. Some machines will allow a given move to be specified in a couple of different ways. The option words enable Tape-to-Shape to interpret these different options for defining a move even when they are both used in the same CNC code file. The option words separate each section into sub-sections. In a sub-section the variable words used to describe the move follow the option word that defines one method for how the move may be defined. Some options will "lock out" another option such that once one of the words under a that option is found, Tape-to-Shape knows which option to use for the definition of the move and will look for variables only under that option for the rest of the block.

The use of the option words shown below is dependent on the section and will be explained later with each section:

```
#OP-STD
#OP-ABS
#OP-INC
#OP-POLAR
#OP-CTR
#OP-RAD
#OP-ANG
#BOS
#EOS
```

Following is an example of a section with option and variable words:

```
@ARC-CW(G02)
X#XPOS
Y#YPOS
#OP-CTR
I#XCTR
J#YCTR
#OP-RAD
R#ARAD
```

@ARC-CW(G02) starts a new section which in this case will create a Clockwise Arc move. The G02 command in the parentheses indicates the CNC command which when found in a block of code will activate this section.

X#XPOS will look for the letter 'X' in the current block of code and assign the numeric value following it to the variable word for the ending X position of the Arc.

Y#YPOS does the same for the Y position.

Note that so far no option words have been used so all arcs interpreted under this section will use the X,Y end position as part of their definition.

#OP-CTR is an option word that begins the sub-section specifying how an Arc would be defined using a specification for the Arc center point.

I#XCTR and J#YCTR give the CNC code letter and variable words for the arc center specification. If one of these words is used in the current block of CNC code, Tape-to-Shape will know to use this option for defining the Arc.

#OP-RAD is an option word that begins the description for the option to define the Arc using the radius input instead of a center specification.

R#ARAD specifies the variable word that will give the Arc radius.

Function words are used in special situations to modify the geometry or values given by earlier words or sections.

Currently there are three available function words:

```
#EVAL()  
#FMT()  
#KILL
```

#EVAL() is used to evaluate an expression. Sometimes the value given for a word in the CNC code does not agree with what is needed for the Shape element to be created.

To operate on the value of a variable word put an evaluation expression on the next line following the use of the variable word in the section. If that variable is found in a block, the evaluation expression is performed. Evaluation expressions are described in detail in the SmartCAM Reference Manual Utilities chapter.

Example: We want the variable word #NHOL to be incremented as it is read in.

```
N#NHOL  
#EVAL(#NHOL=#NHOL+1)
```

Note that this is done only if the word in the previous line is found.

#FMT() will override the format specified in Machine Define. When defining #FMT, you can specify the format to use to interpret the number after the code letter for that instance. If you have the letter V that you want to assign to the following variable word V1, you have to specify the format T3.2 in order for the number after V to be in the format T3.2 and not the default specified in Machine Define. An example of the statement you use to assign this would be:

```
V#FMT(#V1,T3.2)
  Instead of the default V#V1.
```

See the SmartCAM Reference Manual, Utilities, Page 72, for further explanation of the #FMT() function word.

#KILL is used to delete the previous element just created. Sometimes, usually in a custom command section, a point will be made previous to that command in the CNC code file that will actually be performed by the element about to be created. An example of this would be at the start point of a @GRID User\_Cmd.

```
Example: (this is under @CUSTOM...)

@GRID(G36)
#KILL      (remove prior element...Point at corner)
I#INC      (normal definition follows...)
```

Use of variable and option words with specific sections:

- Common     The area between the @SECTION label and the first option word (if used) is considered the "common" sub-section. All variables that are used by more than one option must be placed in this common area. Valid variable words are the same as for the #OP-STD option word.
- #OP-STD     Standard interpretation mode. This is the normal way of defining an element. If no other option is specified this is the assumed interpretation mode. In the standard mode absolute or incremental positions are controlled by separate commands that switch between the active mode. The particular CNC commands are given in the Machine file.
- #OP-ABS     Absolute position mode. Some machines use different CNC code characters to indicate whether a particular word should be interpreted as an absolute or incremental position. This option word will be used in conjunction with the #OP-INC word when absolute or incremental positions may be given in the same block of CNC code.
- #OP-INC     Incremental position mode. This option word is used to indicate incremental position inputs. It is used in conjunction with the #OP-ABS option word.
- NOTE:       All Common, #OP-STD, #OP-ABS or #OP-INC declarations must be made before any other option words are used in a given section. This is because these option words do not "lock-out" other options in the same block. All of the other option words will only search for variable words from that option word sub-section if any of the variable words specified for that option is used.

In the following section by section description of valid variable and option words not all of the listed words are necessary with each section or option. All of the valid words are listed and the requirements of the particular TTS interpreter will determine which ones are actually used.

@RAP section

#OP-STD, #OP-ABS, and #OP-INC options  
#XPOS, #YPOS, #ZPOS -End point position  
#OP-POLAR Polar position option.  
#LNLEN -Length from last position  
#LNANG -Angle from last position

@POS section

#OP-STD, #OP-ABS, and #OP-INC options  
#XPOS, #YPOS, #ZPOS -Set current X,Y,Z position  
#OP-POLAR Polar position option  
#LNLEN -Length from last position  
#LNANG -Angle from last position

@LINE section

#OP-STD, #OP-ABS, and #OP-INC options  
#XST, #YST, #ZST -Start position  
#XPOS, #YPOS, #ZPOS -End point position  
#OP-POLAR Polar position option  
#LNLEN -Line length  
#LNANG -Line angle

@FIXED section

#OP-STD, #OP-ABS, and #OP-INC options  
#XPOS, #YPOS -X,Y hole position  
#ZDPTH -Z depth of hole  
#ZCHK -Z Check height

@ARC-CC and @ARC-CW sections

@XZPLANE, @YZPLANE, and @XYPLANE sub-sections

-define the interpretation of Arcs on orthogonal planes.  
-for use with 3-D Machining, see Machine support chapter, page 6.

#OP-STD, #OP-ABS, and #OP-INC options

#XST, #YST, #ZST            -Start position  
#XPOS, #YPOS, #ZPOS        -End point position

#OP-CTR   Arc by center. An alternate name for STD for arcs.

#XCTR, #YCTR, #ZCTR        -Arc center position,  
Note: see Machine Define items 90 -  
107

#OP-RAD   Arc by radius option

#ARAD                       -Arc radius

#OP-ANG   Arc by angles option

#ARAD                       -Arc radius  
#STANG                      -Start angle  
#EANG, #TANG                -End angle, or Total angle.  
Note: Use only one of these 2  
#XCTR, #YCTR, #ZCTR        -Arc center positions.

Note:            If the XYZ CTR are not specified they must be  
                  calculated using #EVAL. If they are not  
                  calculated Tape-to-Shape will use the last XYZ  
                  positions specified.

@THREAD section

Dependent on Machine Define item #185 Thread type.

#OP-STD, #OP-ABS, and #OP-INC options

#XST, #ZST                   -Start position  
#FTHRD                       -Thread lead

Thread type <0>

#XPOS, #ZPOS                -End point position

Thread type <1> or <2>

#XOV                         -Taper in X  
#XPASS, #ZPASS              -End of thread passes

Thread type <2> only

#V1                         -Total depth of thread  
#V2                         -Depth of 1st pass

@FACE section

#OP-STD, #OP-ABS, and #OP-INC options

#XST, #ZST                   -Start position  
#ZOV                         -Taper in Z  
#XPASS, #ZPASS              -End of thread passes

@TURN section

#OP-STD, #OP-ABS, and #OP-INC options

#XST, #ZST                   -Start position  
#XOV                         -Taper in X  
#XPASS, #ZPASS              -End of thread passes

## @USRV section

A User\_Cmd element is created with an assignment to the template variable word that controls a given machine parameter such as feedrate. This section gives a list of the variable words preceded by the CNC code letter or command. These words may be searched for and assigned out of the current CNC code block, independently of the geometry section which may also be active for the same block. This section is also different from other sections in that a separate User\_Cmd element is created for each variable word found in the current CNC code block.

### Option words:

#BOS        Beginning of span option word. This is the default mode if no option word is used. The User\_Cmd will be created before any move is made in the current CNC code block.

#EOS        End of span option word. The User\_Cmd will be created after any move is made.

Any variable word is valid in this section. All available Template words are described in the SmartCAM Reference Manual, Utilities chapter.

### Example @USRV section:

```
@USRV
F#FEED                ...BOS is assumed, create User_Cmd with
                      #FEED='value' if code letter 'F' is found in
                      current block, before move.
```

```
S#SPEED
#EOS                    ...EOS
D#DWELL
```

## @CUSTOM section

This section allows you to define a custom User\_Cmd to be created when a given CNC command is encountered. It varies from the previous sections in that sub-sections are defined for each custom User\_Cmd using the '@' character followed by the name of the custom User\_Cmd to be created. Refer to the description of User\_Cmd Elements in the SmartCAM reference Manual, Shape chapter.

-----  
TTS specific Machine File Items  
-----

Many of the Machine Define items explained in the SmartCAM Reference Manual, Utilities chapter are used by Tape-to-Shape to interpret CNC code as well as for code generation from SmartCAM's Shape module. It is important that these items be correctly configured for both uses. For example the CNC commands to switch between absolute or incremental positioning modes, as well as commands that control things such as cutter diameter compensation, contour cutting, or punching, are specified by these Machine Define items.

The numeric formats used to interpret the values from the CNC code are also given by Machine Define items. Note that only one format may be supported for a given machine. Some controls allow a CNC program to mix decimal and trailing zero formats for the same positioning word depending on whether or not the decimal point is included. Tape-to-Shape does not support such inconsistent formats.

In addition, there are new Machine Define items specific to Tape-to-Shape. These items were added to the available Machine Define questions when you installed the Tape-to-Shape program. These new items are described below.

3. Tape-to-Shape .TTS filename to use:

Example;  
"NAME.TTS"

This item will specify the TTS interpreter file to be used when you Interpret a CNC code file.

70. TTS, Tool change type:

Choices;  
<0> Change tool automatically with tool number code word  
<1> Wait for a separate toolchange command to activate tool number  
<2> No tool number specified, Use toolchange command to  
index through the Job Plan.

Some machines will automatically change tools when a new tool number is coded. Others will wait for a separate tool change command so the tool number could be given earlier. This is often done with automatic tool changing machines so that the new tool is ready when a change command is given.

71. TTS, Tool code word:

Example;  
"T" Tool number will follow 'T' code word.

Enter the code that will specify the new tool number.

72. TTS, Tool change command:

Example;

"M06" Tool change command will be used if Item #70 = 1

This item specifies the tool change command that indicates that the previously selected tool number is now the active tool.

135. TTS, cutter compensation type:

Choices;

<0> Use cutter comp commands

<1> Use comp vectors

<2> Use sign with Tool width specification for direction and amount

This item specifies the type of cutter compensation to consider in order for Tape-to-Shape to build the original part profile if the part was coded for the tool centerline. Other items such as #120 are also considered.

136. TTS, activate cutter compensation for:

Choices;

<0> Initial compensation move has no offset

<1> Make offset prior to first move

Some controls will activate the offset along with the first feed move while others will make the offset before the move begins.

390. TTS, Control Out:

Example;

"(" All characters following the '(' control out will be ignored

This is the character that indicates the beginning of a comment in the CNC code. It is important so that Tape-to-Shape does not try to interpret the letters in a comment as CNC code.

391. TTS, Control In:

Example;

")" begin using code following the ')' control in command

This is the character that ends a comment. If not used the rest of the block following the Control Out character will be ignored.

392. TTS, Leave space chars in block to be interpreted:

Choices;

<0> No

<1> Yes

Answering No <0>, to this question will strip all spaces from the file being interpreted. Answering Yes <1>, will leave spaces in the code being interpreted. In most cases <0> should be the choice. In the case of a Wiedemann punch press answer <1>

400. Is Rapid positioning command NON Modal:

Choices;

<0> No

<1> Yes

In most cases these CNC commands will be modal. In cases where they are not modal and need to be repeated for this type of move these items should be set to <1>. This applies to items #400 through #407.

401. Is Linear move command NON Modal:

Choices;

<0> No

<1> Yes

402. Are Circular move commands NON Modal:

Choices;

<0> No

<1> Yes

403. Are Arc Center command words NON Modal:

Choices;

<0> No,

<1> Yes, I and J center commands MUST to be repeated.

Note: This only applies when Item #35 is set to 1 for Absolute centers.

404. Are Fixed Cycle Hole operation commands NON Modal:

Choices;

<0> No

<1> Yes

405. Are cutter compensation commands NON Modal:

Choices;

<0> No

<1> Yes

406. Is Absolute positioning command NON Modal:

Choices;

<0> No

<1> Yes

407. Is Incremental positioning command NON Modal:

Choices;

<0> No

<1> Yes

410. TTS, Maximum cutting feedrate:

Example;

"300" Any feedrate over 300 IPM will be interpreted as a rapid move.

Some machines will allow the same move command to be used for Linear cutting moves as for Rapid positioning moves. This item specifies the maximum feedrate for a Linear cutting move. Any move programmed with a feedrate greater than this value will be interpreted as a rapid position move.

411. TTS, Use Contour cutting command for interpretation:

<0> Not required

<1> Use contour On command to distinguish a Linear cutting move from a Rapid move.

<2> Use contour On command to identify Torch tool as active

<3> Use contour On command for both <1> and <2> above

See items #110-113 to specify the Contour On/Off commands.

This item selects how the contour cutting commands will be used to interpret when a Linear cutting move was used instead of a Rapid position move, also it indicates if the use of a contour On command should be used to indicate that a Torch attachment is active instead of the current punch tool. Item #230 controls whether the primary or secondary contour on command is used to indicate a Torch.

Note: Option <2> is an alternative to using the location offset to identify a Torch or Attachment using the @TORCH() TTS section.

412. TTS, Sequence to activate contour cutting command:

Choices;

<0> BOS (Beginning of Span), Before move

<1> EOS (End of Span), After move

This item determines whether to turn the Torch on Before the positioning move or after the move.

-----  
TTS Interpretation sequence  
-----

When a variable or GM command is found in the CNC file, it is cut out of the block being processed, so it is important to know the order of processing. If a variable word is declared in two sections, the section that gets processed first will find it and cut it out. The second section will NOT find it because it was already used by the first. This is often desirable, but it is important to consider when defining TTS interpreter sections

1. The @SKIP section is processed If any of the commands listed in this list is found, the entire block is ignored.
2. If a 3-D Machining file is being created, the command used to select the active orthogonal plane used for Arc commands is checked.
3. Geometry sections are processed and the associated variable words are assigned. An element is created but NOT inserted into the element list.
4. The @USRV section is processed for the #BOS options. This may create and insert User\_Cmd elements.
5. The @CUSTOM section is processed for the #BOS commands. The custom User\_Cmds will be created and inserted.
6. Tool change information is processed. Tool change information is defined by the Machine Define file discussed earlier.
7. The element created in step 3 (if any) is inserted into the element list.
8. The @USRV section is processed for the #EOS options. This may create and insert User\_Cmd elements.
9. The @CUSTOM section is processed for the #EOS commands. The custom User\_Cmds will be created and inserted.

-----  
Hints, Tips, and Tricks  
-----

1. To create a TTS interpreter file from an existing template file (.TMP). Use Edit Plus to edit each applicable template section into the corresponding TTS section. Edit Plus's "split" function using the F8 key when editing is very handy for making the separate lines needed for each word.
2. When testing a new TTS interpreter use the raw mode (condense No) when first testing the results.
3. If you wish to re-code a shape file created using Tape-to-Shape's raw mode (condense No) you may want to create a special code generator separate from the one used on normal shape files. This could take into account the extra elements that are usually created automatically during code generation.
4. CNC Code which is dependant on the order of each code in a block and is separated by a common separator character such as a comma or space (see Machine Define item #392) can be supported by listing the variable words in the section in their correct order.

Example: CNC code-           LINE,x1,y1

Would use TTS section-   @LINE(LINE)  
                              ,#XPOS  
                              ,#YPOS

5. If the CNC commands for a given machine need to include the parenthesis characters "(" you have the option of using other braces such as [] or {} for the section which specifies the command.

Example: @LINE[(0)]

-----  
Common Errors, Trouble Shooting  
-----

1. If incorrect format was specified by Machine Define for a given CNC code word Tape-to-Shape will still attempt to interpret the CNC code file, however the results will usually be very inaccurate.
2. The TTS interpreter file must be matched to the machine file (.SMF) otherwise Tape-to-Shape will give an error for an "Invalid Machine File."
3. Many machine supported by Tape-to-Shape may have some commands and features that can not be interpreted. These are typically special cycles such as mirror image or other repeated geometry features. Tape-to-Shape can be expected to correctly interpret the consistent use of basic geometry commands but machine specific cycles can be supported only as custom User\_Cmds.

-----  
Example TTS sections  
-----

This example section demonstrates the use of option words to correctly interpret both absolute and incremental position commands and the specification of the Arc by either center position or radius.

```
@ARC-CW(G02)
#OP-INC
U#XPOS
W#ZPOS
#OP-ABS
X#XPOS
Z#ZPOS
#OP-CTR
I#XCTR
K#ZCTR
#OP-RAD
R#ARAD
```

This example shows a custom section to support the machine tool's dwell command.

```
@CUSTOM

@DWELL(G04)
P#DWELL
```

MILL.TTS

@RAP(G00)  
X#XPOS  
Y#YPOS  
Z#ZPOS

@RAP(G80=G00)

@LINE(G01)  
X#XPOS  
Y#YPOS  
Z#ZPOS

@ARC-CW(G02)  
X#XPOS  
Y#YPOS  
I#XCTR  
J#YCTR

@XZPLANE  
X#XPOS  
Z#ZPOS  
I#XCTR  
K#ZCTR

@YZPLANE  
Y#YPOS  
Z#ZPOS  
J#YCTR  
K#ZCTR

@ARC-CC(G03=G02)

@RAP(G80=G00)

@RAP(G0=G00)  
@LINE(G1=G01)  
@ARC-CW(G2=G02)  
@ARC-CC(G3=G03)

@FIXED(G81)  
X#XPOS  
Y#YPOS  
Z#ZDPTH  
R#ZCHK

@FIXED(G82=G81)  
@FIXED(G84=G81)  
@FIXED(G86=G81)  
@FIXED(G83=G81)

MILL.TTS continued.

@USRV  
F#FEED  
S#SPEED  
P#DWELL  
Q#PECK

@CUSTOM

@DWELL(G04)  
P#DWELL

LATHE.TTS

@RAP(G00)  
X#XPOS  
Y#YPOS  
Z#ZPOS

@LINE(G01)  
X#XPOS  
Z#ZPOS

@ARC-CW(G02)  
#OP-CTR  
X#XPOS  
Z#ZPOS  
I#XCTR  
K#ZCTR

@ARC-CC(G03=G02)

@THREAD(G33)  
Z#ZPOS  
F#FTHRD

@USRV  
F#FEED  
S#SPEED

PUNCH.TTS

@POS(G72)

X#XPOS

Y#YPOS

@POS(G92=G72)

@LINE(G69)

#OP-POLAR

I#LNLEN

J#LNANG

P#TLWD

C#INDXA

@LINE(G79=G69)

@LINE(G66)

#OP-POLAR

I#LNLEN

J#LNANG

Q#TLEN

P#TLWD

C#INDXA

@ARC-CW(G68)

#OP-ANG

I#ARAD

J#STANG

K#TANG

P#TLWD

C#INDXA

@ARC-CW(G78=G68)

@USRV

C#INDXA

Q#FEED

@SKIP

G93

@CUSTOM

@REPO(G27)

X#XSET

@DWELL(G04)

X#DWELL

@BHC(G26)

I#ARAD

J#STANG

K#NHOL

PUNCH.TTS continued.

@BHC(G29)  
I#ARAD  
J#STANG  
P#INC  
K#NHOL

@LAA(G28)  
#KILL  
I#INC  
J#LNANG  
K#NHOL  
#EVAL(#NHOL=#NHOL+1)

@GRID(G36)  
#KILL  
I#INC  
P#NHOL  
#EVAL(#NHOL=#NHOL+1)  
J#INC2  
K#NHOL2  
#EVAL(#NHOL2=#NHOL2+1)

@GRID(G37)  
#KILL  
I#INC2  
P#NHOL2  
#EVAL(#NHOL2=#NHOL2+1)  
J#INC  
K#NHOL  
#EVAL(#NHOL=#NHOL+1)

-----  
Reviewing CNC programs  
-----

This application will explain how to use Tape-to-Shape to review CNC programs. The example is in a step by step format with screen prompts appearing in boldface text and instructions underlined. A brief description explaining what you are viewing on your display and what Tape-to-Shape will do is presented after the keyboard entry that initiates the action.

The Tape-to-Shape disk included a set of files named TTSLRN, TTSLRN.SH2 and TTSLRN.JSF. for use with SmartCAM V4.0. Sample files named TTSLRN3D, TTSLRN3D.SH3, and TTSLRN3D.JSF are also provided for use with the 3-D Machining module. These files were copied to the DATA subdirectory when you installed the software. If you want to use these files to learn with, enter the filenames provided after "Example:". If you are using the files provided disregard steps 1 and 2 for the purpose of learning but make sure you read and understand the information in steps 1 and 2.

**STEP #1**            Enter the CNC Program

Enter the CNC Program into SmartCAM. The CNC program to be reviewed must exist as an MS DOS text file. If the CNC program is in punched tape format, read the tape using SmartCAM's Utilities - Communicate functions.

**STEP #2**            Create a Job Plan file

A Job Plan is needed when interpreting with Tape-to-Shape. The Job Plan is used to assign the tools and the feeds and speeds associated with these tools.

Enter SmartCAM's Job Plan module and set up the Job Plan for the part you want to Interpret.

Some CNC code will require that you enter the Tools in the order in which they are used to machine the part represented by the CNC program. For information about tool ordering, see the Machine Support chapter, page 16, item #70 <2>

If you are unsure about the content of the CNC program and need to see the part to refresh your memory, we have provided "generic" Job Plans. There is a generic Job Plan for each machine type. Read the Operation chapter of this manual for more information about generic Job Plans. After previewing the part with a generic Job Plan, construct a Job Plan using the correct tools in the order of use. If a tool is used more than once, it must appear in the Job Plan wherever it is to be used. Tape-to-Shape increments through the tools in the Job Plan by 1 each time it interprets a tool change, which is why a tool must appear in the Job Plan file whenever it is to be used.





#### STEP #4 - Viewing the Interpreted Programs

In this step you will view both of the Shape files that were interpreted in step #3.

Select Shape from SmartCAM's Main Menu.

Select Files - Read and press Enter to accept the default filename.

Note that SmartCAM defaulted to the last used filename (RAW).

Displayed on your Screen is the CNC program that was Interpreted in the Raw mode. Note the numerous User Commands and the Small + in the upper right corner of the screen. The + denotes the @END positioning move that is defined in SmartCAM's template files. The User Commands accompany tool changes, speed/feed changes and other User Commands. Note that there are 80 Elements in this file and all the tool Offsets have been set to None. Also note that the geometry displayed is for the tool path centerline.

Select Files - Read and enter the filename for the CNC program that was Interpreted using the Condensed mode (Condense).

Displayed on your screen is the CNC program that was Interpreted in the Condensed mode. Note the only User Commands displayed are when differences occur between the CNC program and the Job Plan, and if you have a User Command not supported by Tape-to-Shape. Also note that there are only 48 elements in the Condensed CNC program and that the geometry displayed is for the part profile. This condensed Shape file contains all the information needed to produce the part for any machine you are ready to generate code for.

Use both the Condensed and Raw Shape files that you Interpreted and View them both to more fully understand the differences between the Interpretation modes.

Select Quit - Tool\_Path - Follow.

Follow shows data for one element at a time. Turn your Mouse off if it is on (F9), and walk around the part using the Up and Down arrow keys on your numeric keypad. Each time you Press the Up and Down arrow keys the adjacent element is selected and its information is displayed at the bottom of the screen. After you have walked around the part press Esc.

Select View - Show\_Path

Show\_Path runs in sequence and graphically displays the tool path, it is used to review the program operations. Watch your display and make note of the machining sequence, and the areas covered by the cutting tools. The tools should clear all obstructions. If you would like to adjust the speed of the Show\_Path press the F6 key and select a Path\_Speed between 0 (intermittent) and 9 (fast). It is also possible to adjust the speed by pressing the corresponding # key (0-9), while Show\_Path is running.

## Overview

This application explains how to convert existing CNC programs for use on different machines. These machine to machine translation issues are important for correct code to be generated for the new machine to produce the part accurately. It is not possible to translate information between incompatible machine types. For example you cannot translate mill programs for use on a lathe or punch. Translation between all like machine types with equivalent capabilities should be possible.

## General Information

Interpret the CNC program using Tape-to-Shape's Interpret Mode. The Condensed mode of Interpretation is preferred as this will eliminate unnecessary lines of code. The Condensed mode should be used if cutter compensation commands are on the CNC program being Interpreted.

## Cutter Compensation

It is important that you know whether cutter compensation commands were included in the CNC program that you are converting. You should also know whether the CNC program was coded for tool center or part profile. Tape-to-Shape's condense mode will automatically recreate part profile if cutter compensation commands are included in the CNC program being Interpreted. Otherwise to get part profile from a CNC program that was originally coded using tool center, you will need to use SmartCAM's Tool\_Path - Wall\_Offset to obtain the part profile.

Some controllers activate cutter compensation differently. Cutter compensation for Lead In moves depends on the machine and controller combination. To generate the correct cutter compensation you may need to change the offset on the Lead\_In moves at the start of profiles. Do this in the Shape module.

## Canned Cycles

Special machine cycles other than the three system User Commands supported by SmartCAM will appear as User Command elements. Bolt hole circles, Grids and Line at angle cycles maybe supported as system User Commands. If they can be supported they will appear on your display correctly.

## Tool Changing

When converting CNC programs from machine tools that only have manual tool changes programmed, to machines with automatic tool changing capability, you will need to Erase the tool change Points. If you are converting CNC programs from automatic to manual tool changing, the tool change points will have to be created. Create the points at the manual tool change position. The Edits to the CNC program can be done graphically using SmartCAM's Shape module.

## Punch Press Conversion

Punch machines that do not have nibble cycle capabilities will have Interpreted Shape files that have lots of Points. There will be a point for each individual hit used to nibble the part. When converting to a Punch with nibble capabilities there are two options.

1. Re-code the CNC program for the new machine using the many individual hits. The new machine's nibble cycle will not be used.
2. Use these points as a guide to recreate the geometry and utilize the new machine's nibble cycle.

NOTE: If you are converting from a Punch with Nibble cycle capability to one without individual hits will be generated automatically.

-----  
Creating CAD data  
-----

This application explains how to transfer data from the existing part geometry data base to a CAD system. This capability allows you to document or update engineering drawings from proven CNC programs. The ability to do this automatically, decreases the chance for human error as well as speeding up the process. Translations to a CAD system require that you have a CAM Connection for your particular system. If you do not have a CAM Connection and would like to purchase one, contact your SmartCAM dealer.

When implementing CAD, CNC code files created either using SmartCAM or by some other means, represent a significant investment in creating a database of part geometry and successful manufacturing processes. Tape-to-Shape is designed to allow the user to re-incorporate this data back into the computer aided design and manufacturing process. By adding dimensions to the geometry transferred from CNC programs to CAD data you have accurate drawings that match the parts you are producing.

#### General Information

Interpret the CNC program using Tape-to-Shape's Interpret function. The Condensed mode of Interpretation is the preferred mode as this will give you the programmed part profile and not the programmed tool center. If you do Interpret using the Raw mode, or the cutter compensation commands were not available to automatically interpret part profile. You will have to use SmartCAM's Tool\_Path - Wall\_Offset to get the needed part profile.

Punch machines that do not have nibble cycle capabilities will have Interpreted Shape files that have lots of Points. There will be a point for each individual hit used to nibble the part. Use these points as a guide to recreate the geometry for the part profile.

View the Interpreted program using Shape's View functions to double check the geometry, before attempting to create CAD data.

Edit the program in Shape and delete all unwanted entities such as Lead In/Out moves and material removal elements like Roughing passes. All non part geometry should be deleted or they will end up on the CAD drawing.

When using CAM Connection to translate CNC programs it is important to specify the layers that you want the information transferred to in the CAD system. CAD layers are specified in the Job Plan using the CAD column to set the layer assignment.

Tape-to-Shape provides educational institutions the ability to utilize the equipment they have purchased, or had donated to them. This will allow students to learn CNC programming using the latest programming technology (SmartCAM) while utilizing the equipment available. This suggested curriculum outline is intended to guide you in setting up a program, tailor it to fit your needs and equipment.

Objective:

To teach N.C. programming using SmartCAM and Tape-to-Shape to produce, edit and verify the code needed to make a part using C.N.C. code. These exercises will also teach keyboarding and word processing skills using SmartCAM's Edit Plus module. An understanding of computers and their use in industry will also be developed using the Job Plan and Shape modules of SmartCAM.

1. Cartesian Coordinates.
2. Absolute Programming.
3. Incremental Programming.
4. Movement Commands.
5. Use T.T.S to program simple box in both Abs. and Inc.
6. Tool Offset.
7. Cutter Compensation and misc. codes, use T.T.S. for user commands.
8. Use T.T.S. to demonstrate the difference between Tool Offset and Cutter Comp.
9. Trig. review for tangency and cutter comp. use T.T.S. to verify.
10. Drilling cycles use T.T.S. to verify.
11. Different types of RS274 standards. (Bendix, Cinn.Mili., Fanuc)
12. Lathe Programming.
13. Advanced Lathe Programming. (Tapping, Threading and Grooving)
14. Punch Press Programming.
15. Advanced Punch Programming.
16. Code for School machine type and produce the part.