

TABLE OF CONTENTS

INTRODUCTION	I
DESCRIPTION OF THIS MANUAL	II
ORIGIN OF THE NAME “SODICK”	III
COMPANY PROFILE	IV

CHAPTER 1 DESCRIPTION OF THE EDM PROCESS **1-1**

GENERAL EDM FACTORS	1-1
WIRE DIAMETER AND WIRE GUIDES	1-1
WIRE TYPE	1-2
RECOMMENDED WIRES	1-2
HARD BRASS WIRE	1-2
FLUSHING	1-3
WIRE SPEED	1-3
WIRE TENSION	1-4
WATER RESISTIVITY	1-4
WORKPIECE MATERIAL AND HARDNESS	1-5
SPECIFIC PARAMETERS OF THE EDM PROCESS	1-5
OSCILLOSCOPE DISPLAY OF 3 DISCHARGE PULSES	1-6
ON	1-6
OFF	1-7
MA	1-7
HP	1-8
IP	1-8
V	1-9
SV	1-9
SF	1-10
C	1-10
WT	1-11
WS	1-11
WP – WC	1-11
UNDERSTANDING THE CONDITION FILE MANUAL	1-11
SURFACE FINISH EXPECTED	1-12
MORE ON THE EDM PROCESS	1-12
OTHER GENERAL TIPS ABOUT WIRE EDM	1-13
START FROM A START HOLE AND RUN IN SUBMERGE MODE .	1-13
CHECK FOR WIRE VERTICAL ALIGNMENT OFTEN.	1-13
MAKING PUNCHES AND DIES FIT.	1-13

CHAPTER 2 MACHINE LAYOUT DESCRIPTION **2-1**

MACHINE:	2-2
DIELECTRIC SYSTEM:	2-2
CONTROL:	2-2
CONTROL SCREEN (CRT) RELATED KEYS:	2-2
KEYBOARD RELATED BUTTONS:	2-3

PROGRAM ACTIVATION KEYS (NC SIDE ONLY)	2-4
MACHINE RELATED SWITCHES (SWITCH SUB PANEL):	2-5
AWT RELATED SWITCHES	2-5
LAMPS	2-6
AMP - VOLTMETERS	2-7
AMP METER	2-7
ADDITIONAL SWITCHES	2-7
ARROW UP – DOWN SWITCH	2-7
E STOP BUTTON	2-7
HOURLY METER	2-7

CHAPTER 3 PROGRAM ZERO DESCRIPTION **3-1**

HOW TO ASSIGN YOUR PROGRAM ZERO POINT:	3-3
UNDERSTANDING WORK COORDINATE SCREENS	3-4

CHAPTER 4 PROGRAM MAKEUP **4-1**

PREPARATION FOR PROGRAMMING	4-3
1) DO ALL NECESSARY MATH:	4-3
2) DECIDE UPON THE WIRE DIAMETER	4-4
3) CHECK THE CONDITION FILE MANUAL FOR THE NECESSARY CONDITIONS AND OFFSETS	4-5
4) CONSIDER THE BEST START POINT	4-5

CHAPTER 5 THE FOUR KINDS OF WIRE MOTION **5-1**

G00 RAPID MOTION (NO CUTTING)	5-1
G01 - STRAIGHT LINE MOTION (ACTUALLY CUTTING)	5-2
G02 AND G03 - CIRCULAR MOTION	5-3
SPECIFYING THE X AND Y VALUES IN A CIRCULAR MOVE	5-3
SPECIFYING THE RADIUS IN A CIRCULAR MOVE WITH I AND J	5-3
CIRCULAR MOTION USING THE R WORD (CONSTANT RADIUS)	5-5
TAPER CUTTING CONSIDERATIONS WHEN CUTTING AN ARC:	5-5
EXAMPLE PROGRAM SHOWING THREE MOTION TYPES G00, G01, G03	5-6

CHAPTER 6 DESCRIPTION OF PROGRAMMING CODES AND WORDS **6-1**

LIST OF G CODES	6-4
LIST OF T CODES	6-5
LIST OF M CODES	6-6
M CODE DESCRIPTIONS	6-6
ADDITIONAL INFORMATION FOR POST PROCESSOR CREATION	6-7
NC PROGRAM EXAMPLES	6-8
PROGRAM EXAMPLE 1	6-8
PROGRAM EXAMPLE 2	6-9

CHAPTER 7 WIRE OFFSET DESCRIPTION **7-1**

UNDERSTANDING OFFSETS	7-1
THREE ADVANTAGES TO USING OFFSETS	7-1
OFFSET PROGRAMMING	7-1
INITIALIZING OFFSET	7-2
OFFSET VALUE	7-3
CANCELING WIRE RADIUS COMPENSATION	7-3
“APPROACH PATTERN” PARAMETER	7-3
IMPORTANCE OF TEST SQUARES	7-5
TEST CUTTING TO CONFIRM OFFSETS	7-5
INCORPORATING DIE CLEARANCE WITH OFFSETS	7-6
WHEN CLEARANCE IS LARGER THAN THE OFFSET VALUE	7-6
USING A “HEADER” IN THE NC FILE	7-7
ENTERING VALUES FOR THE H CODES OF NEWFILE	7-7
SETTING NEGATIVE OFFSET VALUES DIRECTLY	7-8
ALTERNATE METHOD OF DEFINING OFFSET VALUES	7-9

CHAPTER 8 TAPER CUTTING INFORMATION **8-1**

TAPER CUTTING CONSIDERATIONS:	8-1
TAPERING TO THE TOP OF THE DIE (NO DIE LAND)	8-2
MACHINING A DIE WITH A LAND	8-2
HOW TO PROGRAM TAPER	8-3
INITIALIZING TAPER	8-4
EXAMPLE OF HOW TO PROGRAM TAPER CORRECTLY:	8-5
TAPER SETTINGS IN THE CONTROL	8-7
TABLE TO LIMIT	8-7
TABLE TO UPPER	8-8
TABLE TO LOWER GUIDE	8-8
TABLE TO PROGRAM	8-9
TABLE TO NEXT	8-9
TABLE TO DRAW	8-10
NOTES ABOUT DRY RUN	8-10
PARTIAL DRY RUN	8-10
DRY RUN SPEED	8-10
DRY RUN AND LIMIT SWITCHES	8-10
TAPER SETTING EXAMPLES	8-12
TAPER SETTINGS IN BRIEF – OPERATOR INPUT	8-13
MANUAL Z AXIS MACHINES	8-13
POWER Z AXIS MACHINES	8-13
TAPER SETTINGS IN BRIEF FOR 4 AXIS CUTTING (G74-G141)	8-13
MANUAL Z AXIS MACHINES	8-13
POWER Z AXIS MACHINES	8-13
NOTES CONCERNING TAPER CUTTING	8-14
FLUSH NOZZLES	8-14
WIRE TYPE CONSIDERATIONS	8-14
CONVERTING TO DECIMAL DEGREES	8-14
GRAPHICS COLORS	8-14

CHAPTER 9 PROGRAM FORMATTING **9-1**

PUNCHES	9-1
DIES	9-2
FORM TOOLS	9-3
WORDS INVOLVED IN SUBROUTINES	9-3
SUBROUTINES INVOLVE FOUR PROGRAMMING WORDS:	9-3
PUNCH EXAMPLE	9-4
DIE EXAMPLE	9-6

CHAPTER 10 DESCRIPTION OF OPERATING AIDS **10-1**

Q ROUTINE TECHNIQUES (MACROS)	10-1
WHAT IS A Q ROUTINE?	10-1
MACHINE START-UP Q ROUTINE	10-2
VARIABLES IN Q ROUTINES	10-2
ARITHMETIC IN A Q ROUTINE	10-4
OVAL SHAPED DIE BUTTON EXAMPLE	10-5
G AND M CODES FOR Q ROUTINES	10-7
G83 - Transfer Current Position Into H Register	10-7
COORDINATE SYSTEM SHIFTING (G54-G959)	10-8
EASILY PROGRAMMING MULTIPLE DIE OPENINGS	10-8
USING A COORDINATE SYSTEM TO KEEP TRACK OF THE MACHINE'S POSITION	10-9
USING COORDINATE SYSTEM SHIFTING TO COMPARE MEASUREMENTS	10-10
MIRROR IMAGE AND AXIS EXCHANGE	10-10
SUBROUTINE TECHNIQUES	10-13
SCALING	10-13
AUTOMATIC CORNER ROUNDING	10-14
USING AUTOMATIC CORNER ROUNDING "R WORD"	10-14
AXIS ROTATION	10-16
STRING FILE	10-16
EXAMPLE OF STRING FILE	10-17

CHAPTER 11 PRACTICE EXERCISES **11-1**

PRACTICE EXERCISES CHAPTER ONE	11-1
PRACTICE EXERCISES CHAPTER TWO	11-4
PRACTICE EXERCISES CHAPTER THREE	11-5
PRACTICE EXERCISES CHAPTER FOUR	11-7
PRACTICE EXERCISES CHAPTER FIVE	11-8
PRACTICE EXERCISES CHAPTER SIX	11-10
PRACTICE EXERCISES CHAPTER SEVEN	11-11
PRACTICE EXERCISES CHAPTER EIGHT	11-14
PRACTICE EXERCISES CHAPTER NINE	11-15

CHAPTER 12 MAINTENANCE **12-1**

DISPLAY MAINTENANCE SCREEN	12-1
MAINTENANCE CHECKLIST	12-2
BY THE HOUR CHECKS	12-2

DAILY CHECKS	12-2
WEEKLY CHECKS	12-2
BI-WEEKLY	12-2
MONTHLY	12-2
QUARTERLY	12-2
ANNUALLY	12-3
MAINTENANCE DESCRIPTIONS	12-3
RESISTIVITY METER	12-3
DI BOTTLE:	12-3
WORKTANK AND TABLE:	12-4
WATER FILTERS:	12-4
CARBIDE POWER CONTACTS:	12-4
FLUSH NOZZLES:	12-5
WIRE GUIDES:	12-5
LOWER WIRE ROLLER ASSEMBLY:	12-5
WIRE EJECTION ROLLERS:	12-5
WAY LUBRICATION:	12-6
AUTOMATIC WIRE THREADER:	12-6
WIRE CUTTER:	12-7
LOW AND HIGH FLUSHING KNOBS:	12-7
RST BUTTON:	12-8
WIRE GUIDE ASSEMBLY DRAWINGS	12-9
FIGURE SHOWING LOWER HEAD ASSEMBLY	12-9
FIGURE SHOWING UPPER AWT HEAD ASSEMBLY	12-10
LOWER HEAD ALIGNMENT PROCEDURE	12-10
LOWER HEAD ALIGNMENT:	12-11
AWT ADJUSTMENTS AND TROUBLESHOOTING	12-11
AWT U AND V ALIGNMENT PROCEDURE	12-12
“SET” “MACHINE” SUBMODE SCREEN	12-12
WIRE SPOOL SETTINGS A350WP, A500WP, A600WP	12-16
AWT ELECTRICAL CABINET SETTINGS 350,500 ,600	12-17

CHAPTER 13 SET UP PROCEDURE CHECK LIST **13-1**

CHAPTER 14 QUICK REFERENCE SECTION **14-1**

POWER UP THE MACHINE	14-1
POWER DOWN THE MACHINE	14-1
REHOME THE MACHINE	14-1
MOVE AXIS BY “MDI”	14-2
INSTALL WIRE ON THE MACHINE	14-2
NOTES ABOUT INSTALLING WIRE:	14-2
AWT SPECIAL NOTE	14-3
MANUALLY ADJUST FLUSHING	14-3
EDGE FIND BY USING THE ST KEY	14-3
EDGE FIND (USING G80)	14-5
CHANGING THE AXIS DISPLAY READ ANY NUMBER	14-6
VERTICALLY ALIGN THE WIRE	14-7

CENTERFIND OF A HOLE OR SHAPE	14-8
CORNER FIND ROUTINE	14-10
CLEARING THE MEMORY	14-11
CHANGING THE FILENAME	14-11
TYPING NEW PROGRAMS INTO MEMORY	14-11
SAVING PROGRAMS FROM MEMORY TO USER DISK (WITHOUT HARD DRIVE)	14-11
SAVING PROGRAMS FROM MEMORY TO USER DISK (WITH HARD DRIVE)	14-12
EDITING USER DISK OR HARD DRIVE DIRECTLY	14-12
SAVING PROGRAMS FROM MEMORY TO HARD DRIVE	14-13
MAKING A COPY OF A PROGRAM	14-13
LOADING PROGRAMS FROM USER DISK TO MEMORY (WITHOUT HARD DRIVE)	14-13
LOADING PROGRAMS FROM USER DISK TO MEMORY (WITH A HARD DRIVE)	14-14
LOADING PROGRAMS FROM HARD DRIVE TO MEMORY	14-14
SAVING PROGRAMS FROM HARD DRIVE TO USER DISK	14-14
SAVING PROGRAMS FROM USER DISK TO HARD DRIVE	14-15
DELETING PROGRAMS FROM MEMORY (WITH OR WITHOUT HARD DRIVE)	14-15
DELETING PROGRAMS FROM USER DISK (WITHOUT HARD DRIVE)	14-16
DELETING PROGRAMS FROM USER DISK (WITH HARD DRIVE)	14-16
DELETING PROGRAMS FROM HARD DRIVE	14-16
COPYING A USER DISK TO ANOTHER DISK (WITHOUT HARD DRIVE)	14-17
COPY A USER DISK TO HARD DRIVE OR HARD DRIVE TO A USER DISK	14-17
USING THE EDITOR	14-17
COPYING A SYSTEM DISK (WITHOUT HARD DRIVE)	14-19
INITIALIZING A NEW USER DISKETTE	14-20
RUNNING PROGRAMS FROM THE ACTIVE MEMORY	14-21
RUNNING PROGRAMS FROM DISK (WITHOUT HARD DRIVE)	14-21
DRAWING GRAPHICS	14-21
CHANGING POWER SETTINGS WHILE MACHINING	14-22
SWITCHING BETWEEN MACHINE CONTROL AND "APT"	14-22
COPING FROM DRIVE 1 TO DRIVE 2 (WITH HARDDRIVE AND 2 FLOPPY DRIVES)	14-23
COPYING ALL "APT" FILES FROM THE FLOPPY TO THE HARDDRIVE	14-23
COPY ALL PART PROGRAM FILES FROM THE HARDDRIVE TO FLOPPY DRIVE # 1	14-24

CHAPTER 15 CRT SCREENS **15-1**

SCREEN DEFAULT PARAMETERS	15-1
MAL-MDI SCREEN	15-3
MAL SCREEN	15-3
EDIT, GRAPH SCREEN	15-7
EDIT, GRAPH, SCRIN FLAG INPUT SCREEN	15-9
DISPLAY SCREEN	15-11
SET SCREEN #1	15-13
SET SCREEN #2 (ROLL UP KEY)	15-17
SET, TRAVEL SCREEN #1	15-19

SET, TRAVEL SCREEN #2 (ROLL UP KEY)	15-21
SET, SIR/PTR SCREEN #1	15-23
SET, SIR/PTR SCREEN #2 (ROLL UP KEY)	15-23
RUN SCREEN	15-25

CHAPTER 16 REMOTE CONTROL OPERATION	16-1
--	-------------

SELECTING A JOG SPEED	16-1
TO JOG X OR Y	16-2
TO JOG U OR V	16-2
TO EDGEFIND	16-2
TO CENTERFIND	16-2
TO ZERO X Y OR Z DISPLAY	16-2
TO ZERO U OR V DISPLAY	16-2
WIRE ALIGNMENT	16-2

CHAPTER 17 SEVERAL COMMON ALARMS & MESSAGES	17-1
--	-------------

INTRODUCTION

Welcome To Sodick Inc and to Sodick Wire EDM training!

What is EDM? Contrary to what you may have heard, EDM does not stand for Evil Demon Magic. Electrical Discharge Machining was discovered in the 1940's. The first commercial EDM's appeared on the market in the early 1950's. These were the vertical, or ram type, machines (also known as die-sinking machines or "sinkers"). Wire EDM came along in the late 1960's, with the advent of the first numerical controls and has experienced explosive growth. Wire or sinker EDMs can cut any material that conducts electricity.

The Sodick wire EDM control has two sides, the CNC side and the conversational side. This course will teach you the CNC side and machine operation. Many operation procedures will be practiced in class. By the time you finish this course, you should feel quite comfortable with the machine and should be confident with your abilities.

The CNC side allows you to enter and run CNC programs using M, T and G codes. If you have a CAM (Computer Aided Manufacturing) system, your CAM system will be outputting data in a way that the CNC side can understand. Manually programming this side of the control can be tedious. You must do quite a bit of math, and follow strict formatting rules to make your manually written program work.

The conversational side uses a CNC programming language called APT. APT is a graphical programming language where points, lines and circles are predefined first, then the tool path must be defined and then a NC program is created that the wire EDM machine can run. Because many customers do not use APT (they are programming using CAD\CAM) there are VHS videotapes and training manuals showing how to use it.

As a wire EDM operator, you MUST have a basic understanding of manual programming techniques to fully understand your control. If for no other reason than to gain an appreciation for what a CAD\CAM system can do for you, learning CNC programming techniques furthers your understanding of how the wire EDM operates. But a more important reason for learning the CNC side is that you will find times when it is necessary to edit and modify programs. Also, if you understand some of the advanced CNC side programming techniques (like Q routines), you can dramatically shorten the time it takes to create programs.

We wish you the best of luck with the new information that is about to be presented.

DESCRIPTION OF THIS MANUAL

It is the intention of this course and manual to inform beginning and experienced operators of the techniques used in programming and operating Sodick Wire EDM systems. This manual is to be used in conjunction with our class as the course text. This course provides you with good knowledge of machine operation, maintenance and programming.

For experienced operators/programmers, this manual and course should provide you with additional information and technique differences between Sodick and other manufacturers equipment. As experienced operators/programmers know, there are many ways to develop CNC programs and various successful techniques to operate the machine. You can use your own past experience and common sense to develop your own programming and operating style.

This manual is intended for use during the training course and for reference. It is not to be used as a replacement for the manual set supplied with the machine. You should have received the following manuals with your new EDM.

- Control System Volume 3.0A (p.n.6100259)
- Code Instruction Volume 3.0A (p.n.6100298)
- Machine Tool A530D (Or to match your machine)
- Machining Condition
 1. Volume 1.4GL for standard machine (p.n. 6100083)
 2. Volume 3.1 for EFII circuit machines (p.n. 6100236.3)
- APT General
- APT Procedure

It would be wise to familiarize yourself with the above manuals when you return to your shop.

There are many different types of Sodick controls. One of the latest types is the MARK 21 and EX21 controls. These are the two controls discussed in this training course.

It is our sincere hope that this course and manual gives you the information you need to begin operating, programming and maintaining your Sodick wire EDM machine. If you have questions or comments about this manual or operating and programming the Sodick wire EDM machine, please call the **wire applications department** and ask to speak with an engineer. If you feel that something is wrong with the machine and it needs to be repaired, please contact the **service department**. If the machine needs parts, please contact the **parts department**.

		<u>Department manager</u>
Sodick Main Number	(847) 465-0160	
Sodick Service Department:	(847) 465-4473	Mr. Todd Williams
Sodick Parts Department:	(847) 465-4479	Mr. Jesse Lopez
Applications Department:	(847) 465-4434	Mr. Terry Meyer

ORIGIN OF THE NAME “SODICK”

“Sodick” is not an English or Japanese word, rather an acronym. It is a combination of four Japanese characters and was created by president, Toshihiko Furukawa. The name “SODICK” is based on the belief that “everyone should be creative and take initiatives to overcome hardships”. The conception of the name follows.

- | | |
|-------------|--|
| “SO” | The first two letters of the Japanese word “ <u>S</u> ouzu” which translates into creation. |
| “DI” | The first two letters of the Japanese word “ <u>D</u> ikkou” which means to carry out develop, or take action. |
| “C” | The first letter of the Japanese word “ <u>C</u> urro” which means hardships. |
| “K” | The first letter of the Japanese word “ <u>K</u> okufuki” which means to overcome. |

COMPANY PROFILE

August 1976 Incorporated as Sodick Co.,Ltd. with capital of 20 million yen.

Toshihiko Furukawa appointed as president.

December 1976 First CNC EDM system with microcomputer delivered.

December 1978 Relocated corporate headquarters to 1120 Nippa-cho, Kohoku-ku, Yokohama, Kanagawa 223, Japan.

April 1979 Sales of FINE Sodick series CNC EDM systems begun.

October 1979 FINE Sodick 27NC EDM introduced at the 3rd EMO MILANO show in Milano, Italy.

June 1980 Fukui Plant completed as the main production center. Sales begun in U.S.A.

September 1980 First participation in the IMTS'80 show in Chicago, U.S.A.

October 1980 Ultra-fine-finish PIKA-1 circuit for mirror surface finish completed.

(Received [1981 Inventor's Prize] from the Japan Inventor's Association.)

Capital increased to 60 million yen.

December 1980 Sales begun in Southeast Asia region.

January 1981 Sales begun of the 330W CNC wire-cut EDM systems with simultaneous 5-axis control.

May 1981 Sales begun in Australia.

June 1981 Capital increased to 100 million yen

.October 1981 Sales begun in West Germany (at present; Germany)

Shipments of the FINE Sodick series CNC EDM systems reach 1000 units.

December 1981 Capital increased to 160 million yen.

Sales begun in the United Kingdom.

January 1982 Development and introduction of the 8133 EDM power supply unit with built-in 16-bit microcomputer

(renamed the Mark $\frac{1}{2}$ V in April 1982). March 1982 Sodick Ltd. Incorporated in Los Angeles, U.S.A.

First participation in the WESTEC'81 show in Los Angeles.

Developed and begun sales of the high-performance VITOL special machining fluid EDM systems.

(Received [1982 Inventor's Design Prize] from the Japan Inventor's Association.)

April 1982 Sales begun of the FINE Sodick A series CNC die-sinking EDM systems.

August 1982 Capital increased to 180 million yen. October 1982 Completed new corporate headquarters in Shin-Yokohama.

March 1983 New Mark IV and Mark V Series CNC power supply units announced.

South Europe GmbH is established in Frankfurt, West Germany.

November 1983 [1983 Machine Promotion Association Prize] received from the Japan Machine Promotion Association for development of an electric discharge machining system with 4-axis and spindle numerical control.

December 1983 Merged Sodick Inc. and Sodick Ltd. in the U.S. to form Sodick Inc.

September 1984 Developed and announced the High speed small hole drilling EDM.

November 1984 Announced the PICA-2 ultra-fine-finish circuit for wire cutting.
Announced the AIC type CNC EDM system and the CNC power supply unit Mark VII.
February 1985 EPOC series announced in Tokyo and Osaka.
July 1985 Announced of the A5W (later named the H5) high-speed die-sinking EDM system, using VITOL QL (Dielectric fluid).
November 1985 Introduced Mark X and XI series Gold-type numerical control systems.
February 1986 Sodick listed on the second section of the Tokyo Stock Exchange.
October 1986 Announcement of the DiPro X automatic die-designing system and the APM1 CNC electrode forming machine (for domestic customers).
March 1987 Complexion of Kaga plant.
August 1987 Announcement of A Series CNC wire-cut EDM and Mark X and XI series Gold-type numerical control systems.
October 1987 German Mark-denominated corporate bonds issued.
December 1987 Ceramics production line completed at the Kaga Plant.
March 1988 Construction of the Technical and Training Center started in Yokohama.
Expanded Sodick Europe GmbH office at Frankfurt am Main in Germany. April 1988 Announced the GT5 type CNC graphite milling machine and AP1R type CNC EDM.
June 1988 Signed agreement with Japan ASIA Investment Co.,Ltd.(JAIC) to establish Sodick (Thailand) Co.,Ltd. in Thailand.
October 1988 Announced the AP330 type CNC wire-cut type CNC electrode milling machine and the MARK20 series power supply unit with built-in 32-bit microcomputer. CNC wire-cut EDM A350 and A500 selected by the Ministry of International Trade and Industry (MITI) as Good Design products for 1988 in the industrial equipment.
November 1988 Established Sodick (Thailand) Co.,Ltd. Capital in Thailand.
February 1989 Construction of the Sodick (Thailand) Co.,Ltd plant at Navanakorn Industrial Estate in Thailand.
March 1989 Technical and Training Center completed In Yokohama.
August 1989 Ceramics R&D, expanded facilities in Kaga factory.
October 1989 Announcement of entering Injection Molding Machine manufacturing.
November 1989 Grand Opening of Technical and Training Center in Yokohama.
TR Division facilities completed in Kaga factory. Established Taipei branch office in Taiwan R.O.C
.March 1990 The first machine A280L delivery to Europe form Sodick (Thailand) Co., Ltd.
May 1990 Announced the GT10 type large size CNC graphite milling machine and the AH5 type CNC EDM.
April 1991 Japax Inc. became a group company of Sodick.
February 1992 Established 5 subsidiary companies. (Sodick Technical Service Co.,Ltd., Sodick Engineering Co., Ltd., Sodick Mechatech Co.,Ltd., Sodick Plastech Ci.,Ltd., and Sodick New material Co., Ltd.)
March 1993 Announcement of NF series CNC power supply units for die-sinking EDM systems eliminating CNC codes.
October 1993 The good will of Sodick Inc. and that of KGK International Corp, in the U.S. integrated into one.

November 1993 Production begun of the Mold Maker 5 die-sinking EDM systems and A530D wire-cut EDM systems in Sodick (Thailand) Co., Ltd.

March 1994 Development of PGW series wire-cut EDM systems capable of mirror finishing for the first in the world.

April 1994 Established Sodick Singapore Pte.,Ltd. in Singapore.

November 1994 Established Beijing Sodick Machinery-Electronics Co.,Ltd.in China.

December 1994 Established Suzhou Sodick Special Equipment Co.,Ltd. in China.

March 1995 Development of PGM series die-sinking EDM systems.

January 1997 Relorated corporate headquarters to 3-12-1, Nakamachidai, Tsuzuki-ku, Yokohama,Kanagawa.Established Sodick (Taiwan) Co.,Ltd. in Taiwan R.O.C.

April 1997 Established Sodick Electromechanical (Shanghai) Co.,Ltd. in China.

July 1997 Announced to produce the 450MC type Machining Center.

September 1997 Established Sodick (H.K.) Co., Ltd. in Hong Kong.

October 1997 Acquired "ISO9002" for Kaga and Fukui plant from Japan Quality Assurance organization.

CHAPTER 1 DESCRIPTION OF THE EDM PROCESS

The first key concept that you must understand is the basic principle of the EDM process.

One good analogy of what occurs in the EDM process is what is constantly happening in your automobile engine. The spark plug that ignites the fuel in the combustion chamber is constantly receiving a great amount of voltage and current to form a spark. As time goes on, and the spark plug wears, you will notice a certain amount of pitting on the electrodes of a worn spark plug. This pitting is caused by exactly the same elements that work in the EDM process. The heat generated by the spark causes a small amount of the electrode to melt, and as time goes on the melted material is blown away from the electrode of the spark plug. Of course, the spark plug is designed to fight against this pitting, but in the EDM process the melting of material and the flushing away of the melted particles is desirable, and the machine is designed accordingly.

In simplified terms, this is what happens during the wire EDM process. The wire passing through the machine has an intermittent voltage applied to it. As the wire comes close to the material (but not actually touching it), an ionization channel forms and amperage flows through this channel vaporizing a tiny area of the workpiece. This channel or gap must be filled with a dielectric fluid, in this case deionized water that acts as a flushing medium to remove the vaporized particles, an insulator and a cooling medium. The electricity going to the wire is then turned off and the vaporized material will instantaneously solidify to form a small particle. The flushing will then remove the particle from the gap and the process starts over again. This process repeats thousands of times per second. It is this constant process that eventually machines the workpiece to your specifications.

GENERAL EDM FACTORS

WIRE DIAMETER AND WIRE GUIDES

The first consideration the operator should give thought to is the diameter of the wire used to machine the part with. Generally speaking, the larger the diameter, the faster the machining can occur. If the maximum inside radius specified on the print was .005" you would have to use .008" or smaller diameter wire. When you take the (over-burn) the arc distance between the wire and the workpiece plus the radius of the wire, this value will be the radius you will get in the corner.

Sodick allows the following wire diameters and wire guides sizes, on most machines.

.004 Diameter Wire (.11mm or .12mm wire guide)

.006 Diameter Wire (.17mm or .18mm wire guide)
.008 Diameter Wire (.21mm or .22mm wire guide)
.010 Diameter Wire (.26 mm or .27mm wire guide)
.012 Diameter Wire (.31 mm or .32mm wire guide)

NOTE: There are 2 wire guide sizes for each wire diameter. The first size listed is for ultra-precise machining and the second size is for precise machining.

The upper and lower wire guides must match the wire diameter choice. When you change wire diameters, you change the upper, and lower wire guides and also the pre-guide on the threader.

AWT Upper Guide

On machines with AWT, the standard upper guide is made of *sapphire* and is a split design. Sizes are etched in Japanese. When installing these guides, make sure the large lead-in taper is facing up and the guide with the steel insert is the rear guide. There is also available as an option a split *diamond* guide. The front half of these split guide moves in and out to allow the AWT pipe to pass through the upper head when threading the wire. The front half of this guide can be removed and reinstalled in the middle of a job and the wire will still be vertical and in true location.

The lower wire guide is a one piece diamond guide. It is a self aligning screw on type guide. This guide can be removed and reinstalled in the middle of a job and the wire will still be vertical and in true location. Sizes are stamped in metric (see page 12-5).

WIRE TYPE

The second consideration is what type of wire to use. Sodick has developed the power supply to cut at satisfactory speed with plain brass wire. Stratified and coated wires are needed to cut at maximum speed. **Hard brass** wire works best for all general cutting needs and for reliable AWT operation. Use **half hard** wire when taper cutting 10° to 15°, and **soft** wire when cutting 16° or greater. Also look for a wire brand that stays straight when unwound from the wire spool, this will give the best AWT operation.

Spool sizes the machine uses are the **P-5, 11Lb** or **P-10, 22Lb**.

RECOMMENDED WIRES

HARD BRASS WIRE

Sodick KHS Wire: Available from the Sodick Parts Dept.

Specialty Wires

Gisco Mega Cut D: Diffusion annealed speed cutting wire (up to 20% faster than plain brass)

Gisco Mega Cut W: Very soft wire, use for 22° or larger taper cuts.

Gisco Mega Cut A: Use for .004" and .006" diameters

FLUSHING

This is the flow of water that surrounds the wire during the EDM process. The same flushing nozzles are mounted (one around the upper wire guide and the other around the lower wire guide) on machines without AWT. If you have an AWT, the upper flushing nozzle screws onto the upper head. Set the upper flushing nozzle between .005 to .010 inch above the top of the part. The lower flushing nozzle fits onto the lower head and slides up to about .005" inch below the table automatically when the flushing is turned on. This is considered to be perfect flushing conditions. The standard flush nozzle is 6mm I.D. and can be used to a maximum taper angle of 12°. The 10mm nozzle is for taper angles up to 20°, and the 12mm nozzle is for larger angles.

Flushing is one of the most important factors that contribute to the EDM process. This is because the small particles created by the EDM process **MUST** be removed from the work area, otherwise double burning of these particles will occur and the insulation properties of the dielectric are eliminated resulting in unstable cutting. Submerge machining improves the efficiency of the flushing and creates thermal stability and makes for a cleaner atmosphere for the operator. You will find that the better the flushing, the faster and more stable the EDM process will occur and wire breakage will decrease. During rough cutting, you usually want high flushing pressure (T84 code) to remove the particles. During skim cuts, you want low flushing pressure (T85 code) to avoid having the pressure of the flushing deflect the wire. **NOTE:** If the machine is a non AWT, the flushing may remain on when the control stops at an M00 or M01.

When you enter the cut from the edge of a workpiece or when machining close to an internal shape, a portion of flushing is being dispersed by the wall of the part itself. When the wire is not surrounded on both sides of the wire by about .25" of material the flushing is not perfect and the cut may become unstable and the power may have to be reduced.

Flushing mode can be programmed as part of the NC program (T84 high or T85 low) and the operator can take control of the flushing flow by adjusting the two flow valves near the front of the machine or the low and high flush potentiometers below the resistivity meter.

WIRE SPEED

Wire speed is the rate at which the wire passes through the workpiece and is controlled by the speed of the pinch rollers above the wire disposal bin. You should use a faster wire speed for roughing and a slower wire speed for finishing. When roughing, the wire will usually be under a great deal of power and stress as we try to melt the material away as quickly as possible. This also results in pitting on the wire and reduction of the wire diameter you can reduce wire breakage by increasing the wire speed.

When finishing, the stress on the wire is very small and therefore the wire speed can be reduced to conserve wire.

The wire speed can be entered into the cutting condition or into the NC program on machines with AWT. This overrides the knob. All machines can adjust wire speed manually on the control panel.

WIRE TENSION

Wire tension is the amount of tension on the wire as it is driven through its cutting motions. You can think of the wire tension as how much “stretching pressure” is applied from the tension roller to the wire ejection rollers. Actually, the tension is caused by the amount of restriction the wire is given by the tension roller at the top of the wire drive area. The pinch rollers just above the wire disposal bin pull the wire from the wire spool and can be adjusted to eliminate wire curl. If the pressure between the rollers is decreased too much, the wire will slip. The tension roller is the rubber roller that the wire is double wrapped around. Generally, you will want a *slightly* lower wire tension during roughing and a higher tension for finishing.

For roughing, this is due to the fact that the wire will be under a great deal of stress due to the high power for roughing. You don’t want to make this stress excessive by having too much tension on the wire, causing wire breaks. Set the knob to 8 for .010" wire (1200 grams).

For finishing, the tension can be increased to insure that the wire is as straight as possible during machining. This insures that the wall of the workpiece will be as straight as possible. Set the knob to 9 for .010" wire (1600 grams).

The wire tension can be entered into the cutting condition or into the NC program on machines with AWT. This overrides the knob. All machines can adjust wire tension manually on the control panel.

WATER RESISTIVITY

The water resistivity affects the EDM process. The water is intended to be a dielectric, meaning that there should be no electric conductivity in the water. However, in real life this is impossible to achieve. There is a bottle containing deionization resin hooked up to the dielectric tank that removes the charged ions from the water thus reducing the conductivity. While this system works very well, there will be some conductivity in all water. Note: Resistivity is the opposite of conductivity. Companies with wire EDMs will use simple tap water for filling the dielectric tank. They depend on the DI bottle to remove any conductivity from the tap water. However, if tap water is used that contains a great deal of iron, this will use up the DI bottle quickly. For this reason, companies will

install another DI bottle on the incoming water line being used by the wire EDM dept. These bottles are available from a water service company like Culligan. High water resistivity allows a focused discharge, which allows the machine to cut fast, stable and reduces rusting of the workpiece. Low water resistivity allows the discharge to leak out of the cutting area, which slows down the cut, the EDM process becomes unstable and parts rust easier. Water additives available from after market suppliers that reduce rusting may decrease the cutting speed and the life of the DI resin. Normal setting for water resistivity is 50,000 ohms to 100,000 ohms per centimeter (5 to 10 on the resistivity meter). Use a setting of 20 when cutting carbide or aluminum. This allows a more focused discharge for carbide and prevents oxidation on aluminum. When the DI bottle cannot maintain a resistivity higher than 3 it must be refilled with new medium. Some water service companies also offer this service. NOTE: If you are cutting using EFII circuitry, the resistivity should be between 5 and 7 for all materials. Clean the probe in the dielectric tank on a regular basis to maintain a valid reading on the meter (see page 12-3).

WORKPIECE MATERIAL AND HARDNESS

Another factor that contributes to the EDM process is the material to be machined. Any material that will conduct electricity can be machined on an EDM machine. However, some materials will cut more easily than others will. A hardened piece of tool steel will cut better than in its soft state. With soft material, the particles formed will usually be larger, and harder to flush.

Conductivity and melting point of a material are the two biggest factors that contribute to how the material cuts. Carbide is considered one of the more difficult materials to EDM because it doesn't conduct electricity very well and it has a very high melting point.

You may want to make a test cut if you are going to be machining a new material that you have never cut before. Sodick application engineers will be able to help you with the specifics of cutting many kinds of materials if you need assistance.

SPECIFIC PARAMETERS OF THE EDM PROCESS

Many factors called parameters control the way the EDM process occurs. What follows is an attempt to explain each of the parameters. For the beginner, remember that we are just introducing each one. Try not to become bogged down by attempting to memorize each one now. Cutting conditions developed by Sodick will usually generate the desired cutting without you having to adjust them. However, as time goes on you will want to get more familiar with each of the parameters so you can easily fine tune for desired results and even create your own cutting conditions.

Each factor directly contributes to machining speed, part finish, and accuracy. Keep in mind that as you modify the parameters that make up a cutting condition to improve one of these objectives, the other two may suffer.

It is important to know if your machine is equipped with the EF2 circuit or not. Some of the parameters discussed have different ranges if using EF2. The parameters act the same, but the setting value would be different between the two machine types.

Following is a graph that shows 3 EDM cycles. Voltage is applied to the wire until an ionization channel forms. The voltage drops and amperage is now flowing between the wire and the workpiece for the length of time that the ON time designates. The process repeats thousands of times per second.

The cutting condition parameters on the following pages can be entered into the NC program if desired. The cutting condition parameters affect how the EDM cycle looks and are discussed on the following pages.

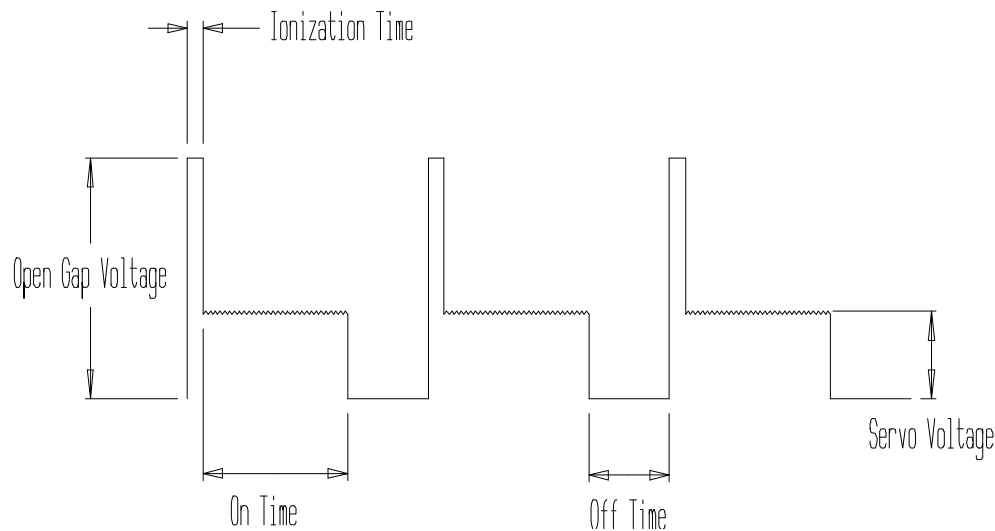


Figure 1 - 1

OSCILLOSCOPE DISPLAY OF 3 DISCHARGE PULSES

Now let's look at the specific "Sodick related" EDM parameters. If you have experience with other brands of wire EDM machines, some of these will be familiar. However, each type of EDM machine will have many things that are specific to that machine. Here we discuss how Sodick does it. NOTE: These can be entered into the NC program if desired.

Example cutting condition :

```
"(      ON OFF IP  HP MA  SV V  SF  C  WT WS WP WC);"  
"C001  =  007 011 117 002  14   018 3 0020 00 000 000 000 000;"
```

ON

Parameter Range: (0-9)

“ON” stands for on time. This parameter controls the length of time that electricity is applied to the wire (per spark). There is a series of numbers that specify the various on times available. The lower the number, the shorter the on time will be.

On time is a factor that can add power to the wire. As you increase the on time, the machine will cut faster but the surface finish gets more porous and accuracy may decrease because of the increased cutting speed. Usually, the on time for roughing is higher than the on time for finishing.

If the on time is too great, the wire will be prone to breakage.

OFF

Parameter Range: (0-32)

“OFF” stands for off time. This parameter controls the length of time that the electricity applied to the wire is turned off between each spark.

Off time is very important since it is during the off time that the particles are flushed out of the gap. Without off time the EDM process could not take place and the wire would eventually short out against the workpiece or break.

Off time is a factor that takes away power from the wire. Increasing the off time will generally mean slower cutting, increased stability and less wire breakage.

Usually on time and off time are considered together for their effects. Off time cannot be lower than ON time.

MA

Parameter Range: (M: 0-9) (A: 0-9)

“MA” affects the adaptive control. It controls the sensitivity and the amount of adaptive control. When the machine’s adaptive control circuitry determines that something is wrong with the cutting (hard spot in material, poor flushing, etc.), it looks to the MA value. MA is a two digit number.

The first digit **M** controls the sensitivity, The lower the first digit, the lower the sensitivity of the adaptive control system. The higher the number, the more sensitivity. This means that a low first digit for MA (1 for example) would tell the control not to be very sensitive to “problems” in the work zone. A setting of 9 would make it very sensitive. A setting of 1 is desired when rough cutting with good flushing.

The second digit **A** controls the multiplier for the off time. When the control senses that something is wrong (based on the M), it will multiply the off time by the value specified in the second digit of MA until the problem is cleared. Then it goes back to its normal setting. We must point out that you don’t actually see the off parameter change, it does

this internally. To obtain maximum speed when rough cutting, lower the A until the cut becomes unstable then raise it back up one number.

For example, say the MA value is 22. The sensitivity would be quite low, and if a problem were determined, the off time would not change dramatically. But, if the MA value is 89, sensitivity would be very high and if a problem were determined, the off time would be increased a great deal until the problem was resolved.

MA is just part of the adaptive control system. If the problem were not overcome with MA, the next stage of adaptive control would kick in. The wire will back up along the programmed path a given amount. If the problem cannot be resolved by this action a short circuit alarm will appear. If the problem is solved, the control would try again in the forward direction.

MA is one of the factors that takes power away from the wire.

HP

Parameter Range: (H : 00-99 P : 0-4)

The H (first two digits) of HP is a high voltage circuit used to increase the open gap voltage. You may find this being used on workpieces thicker than 7 inches, especially for the skim cuts. The A320D machines do not have this circuit because the thickest part that can be cut by the A320D is 7 inches.

The P of HP (last digit) is the output ON TIME if DC arcing occurs. This is used when MA parameter senses the cut is unstable. Increase this value to add more power to the wire to cut faster.

IP

Parameter Range: (000-017) EF2 (000-117)

“IP” stands for Current Peak (I is the electronic symbol for current). IP controls the amount of amperage (current) that is applied to the wire. The higher the number the greater the current.

IP is one of the factors that adds power to the wire. As with on time, the higher the IP, the faster the machining will take place, but accuracy and finish will suffer.

You will always see in the condition manual that on a rough cut IP will be set to 17. The 1st skim cut IP will be set to 16. Normally this is not adjusted during the cut. IP 16 & 17 is the high speed machining circuitry.

IP 1-15 is used for skim cuts #2 and #3. This amperage is a lot lower and can only remove approximately .001" of material at most.

If the machine is equipped with **EF2**, then the IP setting of 117 or 116 (one hundred value) place the machine into the AC mode during cutting (Electrolysis Free).

V

Parameter Range: (0-3) EF2 (0-9)

“V” Voltage is a factor that adds power to the wire. You can think of voltage as being a crude setting for the power range and “on time” as a way of fine tuning the actual power to the wire. The higher the number, the greater the voltage. As you can probably already see, you should always consider voltage and “on time” together.

Voltage is a function of workpiece thickness. The thicker the workpiece, the greater the voltage can be without fear of wire breakage. This is because a thick part allows the power in the wire to be dispersed over a greater area. Picture the wire pulled tight. If you took a (thin) sharp knife, it would be easy to cut the wire. But now picture taking a (thick) baseball bat. It would not be as easy to cut the wire. This analogy shows the relationship of voltage to the cut. Thin workpieces require a lower voltage, while thick workpieces allow a greater voltage.

While cutting, this parameter is usually not adjusted. The cutting condition file will set the value according to which pass is being made. Usually for the first pass it will be set at 3, and the following skim cuts will use 0 or 1 when not using the EF2 circuit.

For machines with **EF2**, the rough-cut will also be set to 3, but the final skim passes will use a larger value. This is because the EF2 machine uses a fine finish circuit during the final skims, and the voltage level needs more than 3 possible settings.

SV

Parameter Range: (00-31) EF2 (000-255)

“SV” stands for servo voltage. This sets the gap voltage between the wire and the workpiece. Lowering this setting makes this gap smaller. Too low makes the cut unstable because the particles are trapped in the gap. The voltage can be read by the voltmeter on the control. This parameter controls the cutting performance and overburn. Think of this setting as the gas pedal on your car. Pushing the pedal to the floor (smaller sv) causes the machine to try to cut faster, but the engine size has not increased. So this parameter actually doesn’t add power to the wire, but will make the machine try to cut faster when lowered. If set too high, it will slow down the cutting speed.

For machines with **EF2**, the value set is an actual voltage target the machine will try to achieve. If it is set at 25, then the machine will maintain a gap voltage of 25 volts. This method of setting the gap voltage directly allows for finer settings. This is required for fine surface finish cutting.

SF

Parameter Range: (0000-2009)

EF2 (0000-9099)

“SF” stands for servo feed. This sets the maximum feedrate of the table. Most apparent when changing SF is the feed rate. SV and SF are factors that allow you to increase or decrease the cutting speed. By adjusting SV and SF you can keep the control from trying to crowd the workpiece into the wire which would result in unstable cutting. Adjust the SF value to avoid holding the feedrate back if the feedrate set is actually reached. In other words, the SF should always be set faster than the machine would actually be able to cut for a given workpiece.

The left-hand digit (thousands) is a value that selects a modifier to the servo. Please use the value recommended from the cutting condition file for the particular skim pass being made.

The following describes this parameter based on the machine type (EF2 or not).

NON EF2: The lower the number the faster the table will move 0004(faster) vs 0006(slower). The following table shows the maximum feedrate the machine would travel at a given setting (cutting air).

0000 or 2000	1.900” min.
0001 or 2001	1.350” min.
0002 or 2002	.850” min.
0003 or 2003	.620” min.
0004 or 2004	.406” min.
0005 or 2005	.300” min.
0006 or 2006	.198” min.
0007 or 2007	.158” min.
0008 or 2008	.118” min.
0009 or 2009	.078” min.

EF2: The setting value is a direct input of feedrate. The higher the number the faster the feedrate. An example follows.

0004	= 0.040” min.
0015	= 0.150” min.
0026	= 0.260” min.
0500	= 5.000” min.

C

Parameter Range: (00-04)

“C” stands for Capacitance. On the Mark 21, capacitance is no longer necessary, but it can still be used. Capacitance is a way of adding more power to the wire. It is only used during skim cutting on the Mark 21 control.

WT

Parameter Range: (00-15)

“WT” sets the wire tension on machines equipped with an auto wire threader. If the setting is at zero, then the machine uses the knob on the control to set the tension amount. If a value is set in the WT parameter, then it uses that value to set tension (ignoring the knob). Please note, the knob value wouldn’t match the setting you would use for the WT parameter. Refer to the introduction of the cutting condition manual for a comparable setting.

WS

Parameter Range: (00-15)

“WS” sets the wire speed on machines equipped with an auto wire threader. If the setting is at zero, then the machine uses the knob on the control to set the tension amount. If a value is set in the WS parameter, then it uses that value to set wire speed (ignoring the knob). Please note, the knob value wouldn’t match the setting you would use for the WS parameter. Refer to the introduction of the cutting condition manual for a comparable setting.

WP – WC

These parameters are not used presently.

UNDERSTANDING THE CONDITION FILE MANUAL

Before writing a program, you must determine the required cutting conditions and offsets to use by looking in the cutting condition manual. Sodick has done much of the research required to develop cutting conditions and offsets. To get the proper conditions and offsets, all you will have to do is select from list in the cutting condition manual. Using a “C” code and “H” code in your program does this.

The condition manual will give you Sodick’s recommended conditions and offsets that you will need to enter into the program. The “C” codes that the manual makes reference to are loaded into the machines memory when it boots up. The cutting condition file’s name is “COND.DAT”, and can be viewed in the EDIT screen.

To program a part, you must know the conditions and offsets (C and H codes) required to produce the cut to your requirements. The condition manual from Sodick is broken down into four basic categories to help you find the proper conditions and offsets.

- 1) **Wire Diameter.** After you select the wire diameter you wish to use, you will reference the condition manual to find the proper pages for the wire diameter you selected.
- 2) **Workpiece Material.** Under each wire diameter available, there are several materials to choose from.
- 3) **Workpiece Thickness.** Each material listed will allow you to choose from several different workpiece thickness.

4) The **number of passes** you desire around the workpiece. This choice has to do with the accuracy and finish requirements you desire. The more passes made, the better the finish and accuracy. You will notice that if you intend to make only one pass, that there are several choices for “first cutting only” in the manual.

We must point out that it would be impossible for Sodick to document the conditions and offsets necessary to machine every conceivable material. If you find that the material you intend to machine or wire diameter you intend to use is not included in the condition manual, you should contact Sodick to obtain their recommended conditions and offsets.

Note: There is also available “cheat sheets” that take information from the condition manual and condense it into several sheets for operator convenience. These sheets are distributed during the 1 week training course. If the machine has EF II circuitry (AC circuitry) and you want to use it, there is also “cheat sheets” for using EFII.

SURFACE FINISH EXPECTED

The required surface finish will determine the number of passes needed. Here is a chart that gives the general relationship between surface finish and the number of passes required for hardened D2 material.

<u>Standard Machine</u>		<u>Equipped with EFII</u>	
<u>CUT</u>	<u>RMS Micro Inch</u>	<u>CUT</u>	<u>RMS Micro Inch</u>
1	100	1	100
2	70	2	90
3	40	3	15
4	20	4	8

MORE ON THE EDM PROCESS

As you can see, there are many factors that contribute to the EDM process. It would be impossible for beginners to memorize each factor. One of our goals in this class will be to teach you how to determine what cutting condition to use and what parameters are changed most often, then you can fine tune the previously developed conditions to your liking. But you will not be left to develop new conditions on your own. You can reference the Sodick condition manual or call the applications department for help.

Another point to make is that all of these factors work together to form a good cutting condition.

"(= ON OFF IP HP MA SV V SF C WT WS WP WC);"

"C001 = 007 011 117 002 14 018 3 0020 00 000 000 000 000;"

TO CUT FASTER

Raise ON
Lower OFF
Raise P
Lower A

TO CUT SLOWER

Lower ON
Raise OFF
Lower P
Raise A

OTHER GENERAL TIPS ABOUT WIRE EDM

START FROM A START HOLE AND RUN IN SUBMERGE MODE .

You will find that trying to start from an edge is undesirable for two reasons. First the flushing will always be poor when you start from an edge. You may have to lower the power until the wire gets into the workpiece. Second, for hardened workpieces, the stress from heat treatment is different around the edges of the part than it is in the middle of the part and the material may spring. Wire cutting relieves stress in the piece. For these reasons, we recommend always starting from a start hole and never coming within about .500" of the edge of the rough stock.

CHECK FOR WIRE VERTICAL ALIGNMENT OFTEN.

It is a good habit to check the vertical alignment prior to machining a workpiece. You must confirm that the wire is vertical to keep from machining unwanted taper in the part.

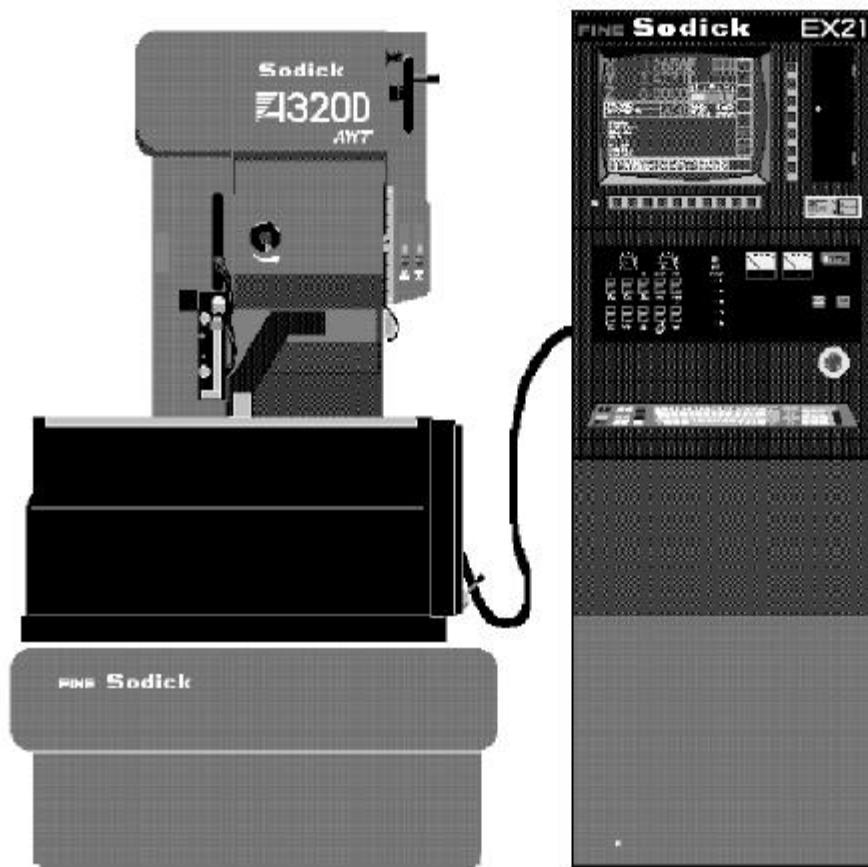
MAKING PUNCHES AND DIES FIT.

One very important consideration when machining punches and dies with small clearance is that a small radius **MUST** be programmed on **ALL** outside square corners of a punch. If it is not programmed, the punch will not fit into the die, due to the small radii generated on the inside corners of the die from the radius of the wire. This is especially true when the amount of die clearance is very small. If the die clearance is quite large, you may get away with not doing this, but the die clearance will not be uniform throughout the punch and die shape.

CHAPTER 2 MACHINE LAYOUT DESCRIPTION

The second key concept to understand is the basic makeup of the machine. Our intention first is to look at the machine from the operator's viewpoint. Later, we will look at the machine from the programmer's viewpoint.

While the configuration may change from model to model, you will find that there are many consistencies from one Sodick wire EDM machine to another. This section is intended to acquaint you with the components of the machine and control. Here is a drawing that shows the A320D, one of the popular machine models:



This model is made up of three basic components; the machine tool, dielectric system, control.

MACHINE:

This picture shows the control, base, work tank, upper arm, automatic wire threader etc.

As the table moves from right to left and vice-versa, it is moving along the “X” axis. As the table moves in and out, it is moving along the “Y” axis. The up and down motion of the upper head is the “Z” axis.

The upper head can also move in two directions parallel to the X and Y-axis. These motion directions are primarily used to create taper. As the upper guide moves left to right, it is moving in the “U” axis. As the upper guide moves in and out, it is moving in the “V” axis. (More on taper later, chapter 8).

Each axis of motion also has a direction implied (plus or minus). We will discuss this a little later.

DIELECTRIC SYSTEM:

The dielectric system is the water-filtering tank located behind or to the side of the machine tool. This tank contains filtering and de-ionizing systems for cleaning the water for use in the Wire EDM machine.

CONTROL:

The control allows you to activate the functions of the machine. Conversational programs are input through the control. CNC programs can be entered, modified, and run through the control. Conditions and offsets can be created and modified. Almost anything that has to be done on the machine begins with the control.

Here is an introduction to many of the most important keys and switches on the control panel. These are important ones to be familiar with. To make it a little easier, we break them down into logical categories.

CONTROL SCREEN (CRT) RELATED KEYS:

Soft Keys - The soft keys are the horizontal and vertical series of buttons that are below and to the right of the control screen. They are NEVER used when inputting a conversational (APT) program. They are used on the NC side to activate the various control screens like Manual Mode, Edit, Set, Run, etc. and to control the sub-modes of each major mode. When used, the control screen itself will actually show you the function of each soft key, so you can easily tell what each key does at any particular time.

NOTE: the bottom soft key of the vertical row will always bring you to the run screen although the monitor does not appear to show that.

KEYBOARD RELATED BUTTONS:

F1 through F10 (function keys) - The F1 key through F4 keys will show you information about how the present job is set up. Use the ESC key to turn off these sub-screens or press the key again.

F1 shows axis positions, speed, offset, taper, C cond., Date, time, filename, etc F2 shows mirror state, rotation amounts, AWT retry settings, etc. F3 is blank, but use this button to cancel the screen saver if screen saver is on F4 shows the work coordinate screens G54 to G59. Press again to see the next level of coordinates G154 to G159. Again for the next level — etc.

These keys have different functions if you are programming with APT.

ESC Key - (Escape) If pressed while drawing graphics, graphics will stop. It also stops the search and replace functions in the edit screen.

HOME CLR Key: When in edit, you can use this key to move the cursor to the top or bottom of the file you are looking at. Select the direction by first pressing the arrow key (up or down) then press the Home CLR key.

BS Key - Back space key. Use it to delete characters to the left of the cursor when in edit

ROLL UP Key - When in the edit screen pressing this key causes the NC file to roll up 1 complete page.

ROLL DOWN Key - When in the edit screen, pressing this key causes the NC file to roll down 1 complete page.

SHIFT Key - When in edit, press and hold the shift key along with one of the number keys to get the extended character set, () \$ % < > keys. It can also be used to move the yellow information box (F1 to F4 information keys) to the bottom of the screen.

RETURN Key - It is used on the NC side in the MAL, MDI or EDIT mode to signify the end of a program line.

ALPHABET Keys - These keys are used to input the various letters on either the NC side or conversational side.

NUMBER Keys - These keys are used to input the various numbers on either the NC side or conversational side.

ARROW Keys - The arrow keys allow you to move the cursor on the control screen. This aids in the various times when the control is giving you a list of information to choose from.

INS Key - When in edit this toggles the cursor between insert and exchange modes.

DEL Key - When in edit this key will delete the character the cursor is highlighting.

MFR - This stands for Manual Feed Rate. This four position switch allows you to select how fast you want the manual axis jog rate to be and also the dry-run feedrate. "0" is the fastest feedrate and "3" is the slowest feedrate.

U/V On - EX21 controls do not let you move the U or V axis without first pressing and holding down this key while manually jogging the U or V axis. MARK21 controls have a UVW key that stays on when pressed once.

X, Y, U, V, Z, plus and minus - These buttons allow you to manually move the axes on the machine in the desired directions (plus or minus). The motion rate will be set by MFR (Manual Feed Rate).

ST - This stands for Sensor Touch. This allows the operator to edge find by jogging the wire into the workpiece until the wire is flush with the part. NOTE: There is a more accurate way of edge finding using the G80 code. See page 14-3 for details about ST.

PROGRAM ACTIVATION KEYS (NC SIDE ONLY)

ACK - Means acknowledge. Whenever the control generates an alarm message on the control screen and the buzzer goes off, this key must be pressed to acknowledge the fact that you understand the message.

HALT - This key can be pressed to temporarily stop the actual running of a program.

RST - Means restart. After halting or when the control automatically stops the program's operation maybe due to wire breakage, M00 or M01 this key allows you to continue from where the machine stopped.

OFF - This key will stop the program permanently. There will be no way to continue without going back to the start point and running the program again. You must be careful when using this key, since you will not get a "second chance". The machine will just stop. We recommend that you get in the habit of ALWAYS pressing the HALT key to stop the activation of a program. Then, if you still want to completely stop the program, press the OFF and ACK key.

ENT - Means enter. For program operation, it activates the program to be run.

MACHINE RELATED SWITCHES (SWITCH SUB PANEL):

- 1) Wire Run Switch - This switch allows you to manually turn on or off the pinch rollers above the wire disposal bin. These rollers pull the wire from the wire spool.
- 2) Flushing Switch - This switch is used to manually turn on and off the flushing pump.
- 3) High Press Switch - This switch is used to manually turn on and off the high pressure flushing mode.
- 4) Wire Speed - This knob allows you to manually adjust the speed of the wire as it passes through the machine and workpiece. If wire speed is set too low when roughing, it may cause wire breakage. For finishing passes, you can turn down the wire speed to conserve wire.
- 5) Wire Tension - This knob allows you to manually adjust the wire tension. If you set this too low during a cut, it may cause inaccuracy on the workpiece. If you set this too high on a roughing pass, it may cause wire breakage.
- 6) KYB / REM SWITCH or KB - RC For machines with a remote pendant, this switch selects between the remote or the keyboard MFR setting. If at the KYB position, the remote still works, but the machine moves at the KYB setting.

AWT RELATED SWITCHES

- 7) AWT I or II Wire threader switch. AWT I cuts the wire. AWT II threads the wire.
- 8) WIRE GUIDE - Use this switch to manually open & close the upper wire guide.
- 9) AWT W/ C - W position turns on the AWT water jet . The C position (clutch) frees the AWT pipe so it can slide up and down manually (C is not used on A320D).
- 10) Tub Fill Switch- This turns on the worktank fill pump.
- 11) Drain Open Close Switch - Opens and closes the worktank drain valve.
- 12) Submerged (or non) Switch- This switch causes the machine to switch between submerged and non-submerged mode. NOTE: If the high flush potentiometer is at a low setting the worktank fills up slowly when in submerge mode.

LAMPS

13) UPS Lamp - Shows the status of the battery backup device (A530 & A320 only). During normal operation this lamp is always on. If the incoming power to the shop goes off, this lamp will be on for as long as the battery backup lasts, which is about 15 Minutes. If the power in the shop returns within 15 Minutes, the machine will reboot and continue executing the program. CAUTION: If the control is at an M00 and the power goes off and back on within 15 minutes the machine may continue onto the next line after the M00.

14) Alarm Lamps - These lamps, if on, would indicate that a fuse has blown in the power supply. Do not attempt to continue cutting. Please call the service dept. for exact location of the fuse, as this is different for each machine model

15) Submerged lamp - This LED shows the status of the worktank float switch. On indicates that the worktank is filled up to the floats.

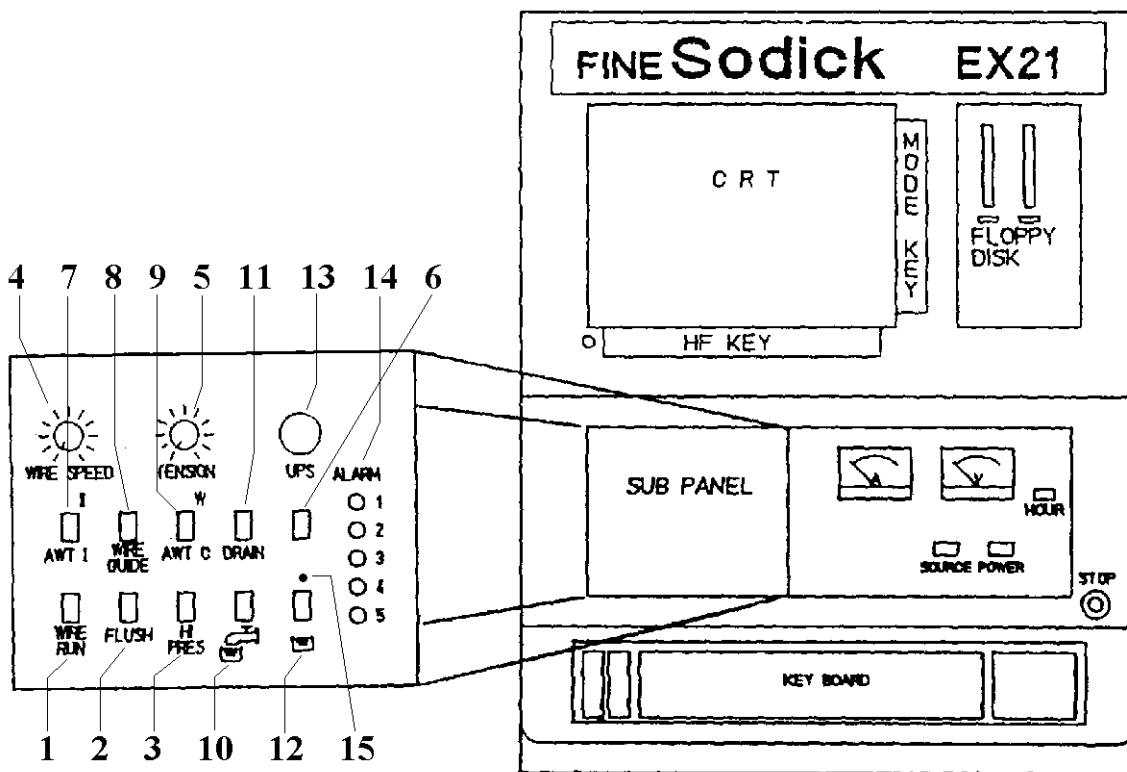


Figure 2-2 EX21 control with AWT

AMP - VOLTMETERS

The Amperage and Voltage meters are located to the right of the Sub-panel. These meters are used during machining to show the status of the power supply.

AMP METER

When the IP is at 16 or 17 read the top scale on the AMP meter (0-40). When the IP is 15 or lower read the lower scale (0-4).

ADDITIONAL SWITCHES

ARROW UP – DOWN SWITCH

On a MARK21 control there is a switch directly to the left of the KB- RC switch that has up and down arrows by it. This switch is not used.

E STOP BUTTON

On the control, there is a large red push button. This button turns off the “POWER” to the machine (all pumps, motors). If it is pressed the “SOURCE” remains on, and the computer is still powered up. DO NOT lean on the table with the power off, as you will lose location. It is also wise to check the negative limits whenever turning the “POWER” switch back on.

To completely shut down the machine, also press and release the “SOURCE” switch.

HOURLY METER

This meter displays the machining hours on the machine. The small switch located on the meter can be used to reset the meter to zero. To reset the meter to zero, slide the switch up and press it in. To lock the reset switch, leave it in the down position.

CHAPTER 3 PROGRAM ZERO DESCRIPTION

Part of the second key concept involves understanding something called “program zero”. This feature allows the programmer to choose a convenient point to work from.

You, as the operator / programmer, have total control of where you would like your program zero point to be. Some EDM operators like to choose the start point of the wire as their program zero point. But a much more logical program zero point would be the point on the part print where all (or most) of the dimensions are coming from. Using this technique, dimensions going into the program will match the dimensions on the print.

When you work from program zero, it is called working in the “absolute” mode of programming. That is, all dimensions come absolutely from program zero. This differs from another style of programming called incremental programming, where each position is taken from the last position of the wire (not from program zero).

Here are two drawings that show how to determine your program zero point:

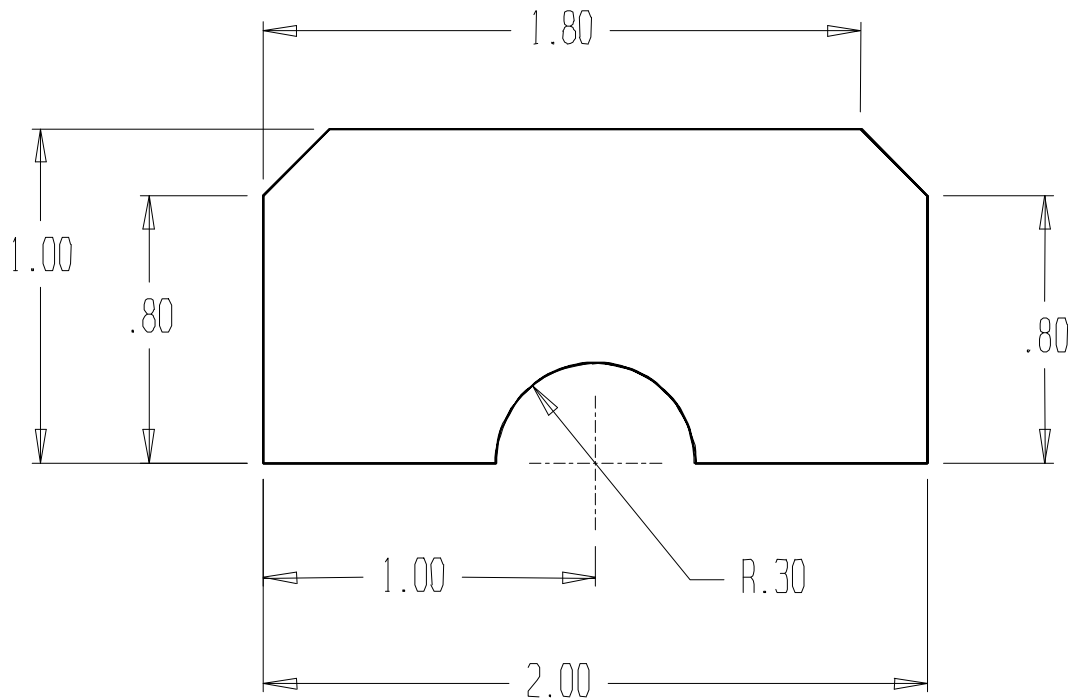
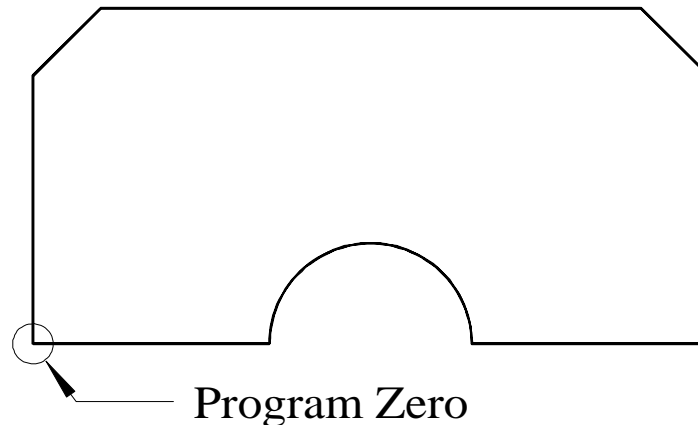


Figure 3-1

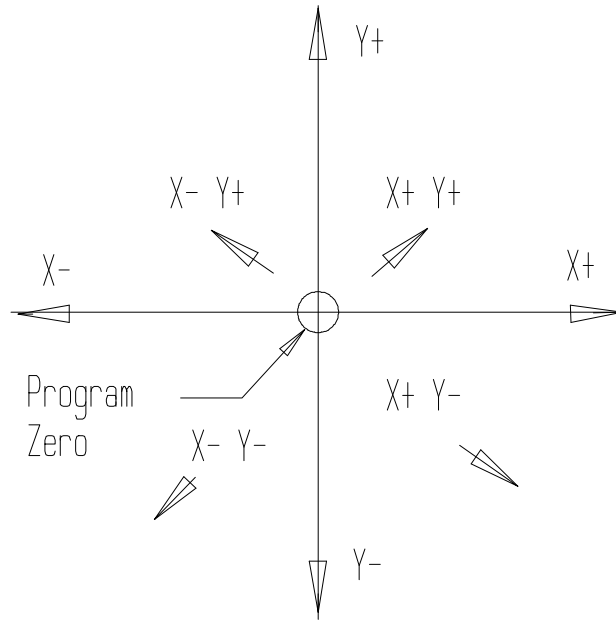
Notice that all dimensions on Figure 3-1 come from the lower left corner of the part. Since this is the case, the lower left hand corner of the part would make an excellent point at which to place the program zero point. Figure 3-2 illustrates this.

Figure 3-2



Also, it is a very good idea to mark up your print to show the location of the program zero point and wire start point so that you can keep track of what you are doing. Another point about program zero: Coordinates that go into your programs will be including a minus direction. The plus sign (+) is assumed, meaning you can omit it when programming positive values. But the minus sign (-) is required for negative coordinates. Positions to the right of program zero in X and above program zero in Y are considered to be plus. Positions to the left of program zero in X and below program zero in Y are considered to be minus (-).

We should point out one thing about determining plus and minus for X and Y. Think about the wire actually moving in X and Y. In reality, the TABLE is actually causing the X and Y motion. This means that to cause the wire to move to the right (X+), the table actually has to move to the left. Think of the WIRE movement and you will not get confused.

*Figure 3-3*

HOW TO ASSIGN YOUR PROGRAM ZERO POINT:

In any wire EDM program, you will tell the control where your program zero point is. To do this, you will be telling the control how far it is from the program zero point to where the start point is. Before running any program, you will always position the wire to its starting point. This may be a position where a start hole is located.

A G92 command is used to assign program zero. You will include the distances and directions from the program zero point to the wire start point in your G92 command. The best way to get the idea is to look at figure 3-4 .

Notice that the program zero point is the lower left hand corner of this part. Also, notice that the wire start point (when the program is activated) is .500 over in X and -.500 down in Y from program zero. Here is the correct command to assign program zero for this example:

N010 G92 X.500 Y-.500

The Y value is minus since the wire starts from a position that is below program zero in Y. If you do not want the G92 in the program, position the machine at the start point location. Go to "MAL", "MDI" and type the value of the start point and press enter to change the CRT displays so the machine knows where it is on the part.

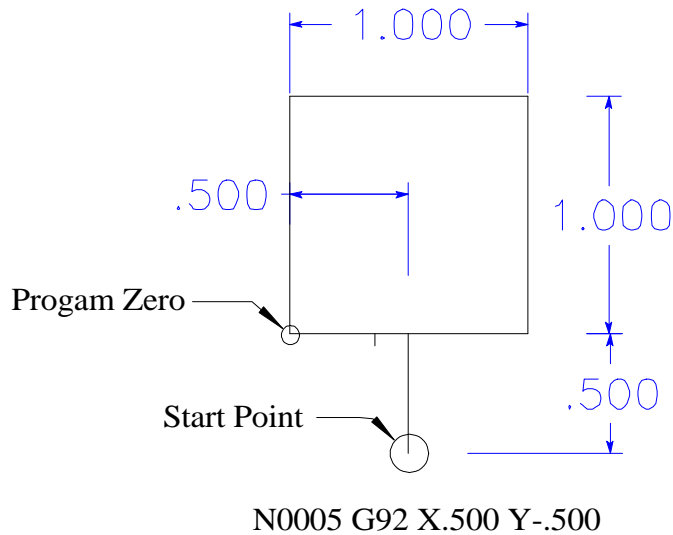


Figure 3-4

UNDERSTANDING WORK COORDINATE SCREENS

Sodick allows you to assign up to sixty (60) different program zero points. This could be helpful when you are machining multiple parts. However, most of the time you will only be interested in one of the available work coordinate screens. You will usually start with coordinate system number zero, which is commanded by a G54. If you are setting up more than one part at the same time or cutting multiple openings in a part, you might be commanding other work coordinate screens other than number zero.

Prior to programming your G92 you can tell the control which coordinate system you want to work in. This means that you can include a G code in the program PRIOR to the G92 command.

Here is a list of all of the coordinate systems available

<u>G CODE</u>	<u>DISPLAY AT MACHINE</u>
G54 to G59	= work coordinates A00 to A05
G154 to G159	= work coordinates A10 to A15
G254 to G259	= work coordinates A20 to A25
G354 to G359	= work coordinates A30 to A35
G454 to G459	= work coordinates A40 to A45
G554 to G559	= work coordinates A50 to A55
G654 to G659	= work coordinates A60 to A65
G754 to G759	= work coordinates A70 to A75
G854 to G859	= work coordinates A80 to A85
G954 to G959	= work coordinates A90 to A95

NOTE: G959 (screen95) cannot be set to a new number as this is displaying the machine coordinates located from the machine home position. The “A” in the above examples means absolute (G90). An “I” would mean incremental (G91).

There may be times when you will want to use the various coordinate systems (other than G54) for some special purposes. For example, lets say you had job #1 on the worktable, and it was already located. If a second job needed to be set up and located you probably would not want to disturb the zero set of job #1. Simply change to a second work coordinate system by entering G55, then locate your second job and set zero in this screen. Now your 1st work coordinate will not have been disturbed. Also be aware of the A(absolute) G90 or I(incremental) G91 ahead of the work coordinate screen number. Figure 3-5 shows us an example.

Understanding WorkCoordinates

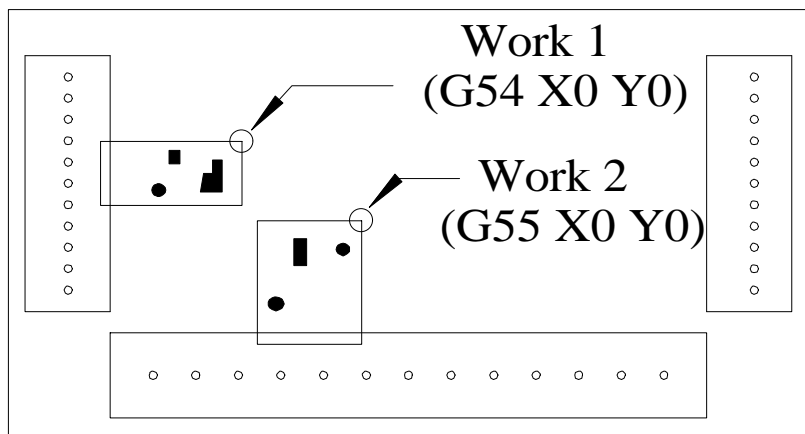


Figure 3-5

When setting up these two parts on the worktable, the operator would use G54 when corner finding workpiece 1. G55 work coordinate would then be used when corner finding workpiece 2.

CHAPTER 4 PROGRAM MAKEUP

Another facet of EDM operation is knowing how the machine will process a program. A CNC program is very similar to giving a set of instructions to a person over the phone. Over the phone, a person must be very explicit, since no hand motions or drawings can be used. A CNC wire EDM machine requires this type of explicit instructions.

A program is made up of commands. Each command includes information to tell the machine what to do. A command may tell the machine to turn on the flushing or move to a particular position or generate a circular movement. A command includes a letter (X, Y, I, J, R, G, N, T, M etc.) and a numerical value.

It is important to remember that any program will be generally executed in sequential order. That is, the control will start from the first command of the program, execute it, then move on to the next command.

For beginning manual programmers, it can be somewhat difficult to tell how much information to include on a line. But as we discuss program formatting (in concept number five), you will find that the recommended program formatting is easy to follow. Also, you won't have to memorize the information in every command. We will be showing you good examples of program format in a way that allows you to simply copy the recommended format. You will find it much easier to "recognize" these commands in format than it is to come up with the proper format off the top of your head. Figure 4-1 is a print to cut a square.

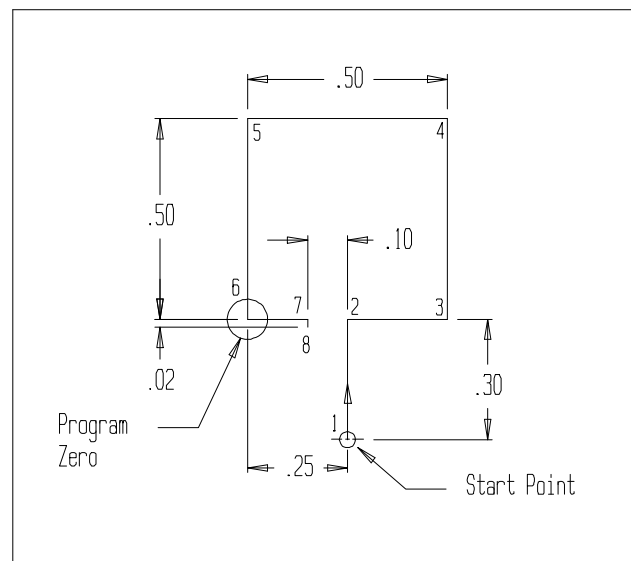


Figure 4-1

Program for figure 4-1

```
N005 G54 (SELECT COORDINATE SYSTEM A00)
N010 G92 X.250 Y-.300 (SET PROGRAM ZERO)
N015 G90 (SELECT ABSOLUTE MODE)
N020 C411 (SELECT CUTTING CONDITION)
N025 G42 H061 (SELECT RIGHT OFFSET AND VALUE)
N030 T94 (SUBMERGE MODE )
N031 T91 ( AUTO WIRE THREAD)
N035 T84 (HIGH PRESSURE FLUSH ING)
N040 G01 Y0 (MOVE UP IN Y)
N045 X.5 (MOVE TO THE RIGHT)
N050 Y.500 (MOVE UP IN Y)
N055 X0 (MOVE TO THE LEFT)
N060 Y0 (MOVE TO PROGRAM ZERO)
N065 X.150 (MOVE TO MOO POINT)
N070 G40 Y-.02 (MOVE TO POINT 8 AND CANCEL OFFSET)
N075 M00 (PROGRAM STOP TO SUPPORT PART)
N080 G01 X.250 (MOVE TO ENTRY LINE)
N090 G01 Y-.300 (MOVE BACK TO START POINT)
N095 M02 (END OF PROGRAM)
```

The information in each command may be foreign to you yet. The control will start executing this program from the command beginning with N005. When it finishes with line number five, it will go on to line number ten, then to line number fifteen, and so on. All programs will be executed sequentially, one command at a time.

NOTE: The information inside parenthesis are messages for you to read. The control ignores the information.

The “N” words are sequence numbers. They are not required to be in the program, unless your using sub routines. However, we recommend that beginners use them in each program and keep them in a logical order for program line identification purposes.

PREPARATION FOR PROGRAMMING

Another part of the second concept: You must prepare to write a manual CNC wire EDM program. This is one of the most important points, even though it has nothing to do with program commands at all. Let us state this again: **YOU MUST PREPARE TO WRITE PROGRAMS!** You know the old saying, “GARBAGE IN/GARBAGE OUT!” It truly applies to CNC wire EDM manual programming.

There are four steps that must be taken before you attempt to write a manual program. If you do these four steps, you will be able to get through the entire program without having to break out of your train of thought. But if you do not take these steps, you will find yourself constantly back-tracking to do something that would have been better and more easily done earlier. Also, for beginning programmers, you will find it easier if you have the “leg work” done up-front. Most companies now program using CAD/CAM.

Here are the four recommended steps to prepare to write your program:

- 1) Do all necessary math
- 2) Decide upon the wire diameter to use
- 3) Check condition file manual for necessary conditions and offsets
- 4) Consider the best “start point” for the wire

Let's discuss each step.

1) DO ALL NECESSARY MATH:

Almost all CNC wire EDM programs require math calculations to be done. In essence, you will be driving the wire through a series of coordinates that are calculated from your program zero point. This is done in a point to point or “dot to dot” method. So, prior to trying to write your program, it is wise to calculate the various positions you will be driving the wire through. These calculations can sometimes get complicated. To calculate the coordinates could involve some trig calculations as well as addition and subtraction.

One logical method to do this involves coming up with a “coordinate sheet” on which you record the coordinates that will be necessary for the program. It is wise to mark up your

print to show the location of your program zero point, and number each location that the wire will pass through. Then, on your coordinate sheet, write down the X and Y coordinates of each point number. Figure 4-2 shows an example of how to use the “coordinate sheet”.

Notice that the programmer knows that this die will include an arc-on and off technique to get a smooth entry and exit so he sketched in the approach movements and included the entry and exit points in the coordinate sheet.

Once the coordinate sheet is filled in, the operator/programmer will be simply copying the coordinates from the coordinate sheet right into the program. This obviously keeps the operator/programmer from having to break out of his or her train of thought while writing the program to come up with coordinates.

Another point to make about the coordinate sheet is that having it truly helps when trying to correct any mistakes which have been made. While verifying the program at the machine (during dry run or graphics), if you find a mistake, it will be easier to correct the mistake if you have a coordinate sheet. Also, it may point in the direction of how the mistake was made so you can improve in the future.

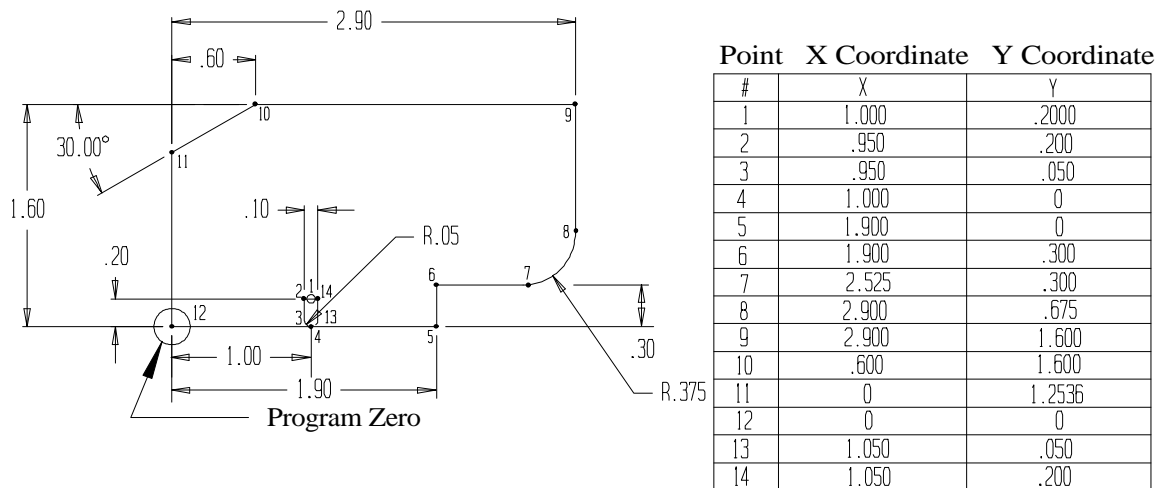


Figure 4-2

2) DECIDE UPON THE WIRE DIAMETER

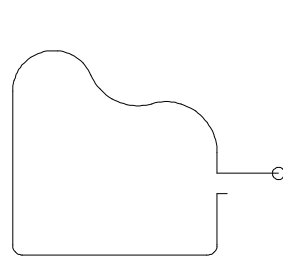
Many times the print will specify a MAXIMUM inside radius allowed for one or more radii. This means that you MUST use a wire size that is SMALLER than the maximum allowed inside radius. For example, if your print specifies that an inside radius be no greater than .005 inch, you may think that you could use .010 diameter wire (since the wire radius is .005 inch). But you would be wrong! You must also consider the “overburn” that is involved in the EDM process. A .010 diameter wire will cut a radius of more than .005 inch (depending on the condition and number of skim passes), so if the .005 inside radius is critical, a wire diameter of .008 inch or less must be used.

3) CHECK THE CONDITION FILE MANUAL FOR THE NECESSARY CONDITIONS AND OFFSETS

Prior to writing your manual program, you must know the conditions and offsets required to produce the part to your requirements. The condition manual from Sodick is broken down into four basic categories as described during chapter one. Review the information presented earlier when deciding which cutting conditions to use (see page 1-11).

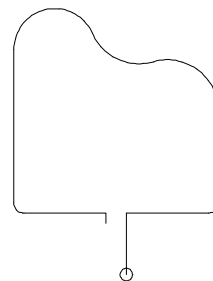
4) CONSIDER THE BEST START POINT

Prior to trying to write your program, it is necessary to study the print to determine a wise starting point for the wire. For punches (or any outside shape), it is important that you select the start point in a way that allows the tab to easily be ground off with a surface grinder. If you are not careful about your start point selection, you may find it difficult to grind the tab area after the punch has been removed from the wire EDM machine.



Good Start Point
Selection

Figure 4-3A



Poor Start Point
Selection

Figure 4-3B

As you can see, the part in Figure 4-3A allows the tab area to easily be ground. The piece could simply be laid flat on the chuck of the surface grinder for the grinding operation. But the part in Figure 4-3B shows a start point that would require an elaborate setup on the surface grinder.

CHAPTER 5 THE FOUR KINDS OF WIRE MOTION

The third key concept to wire EDM programming is that there are four kinds of wire motion. It should be refreshing for beginners to know that there are only four ways to cause the machine to move around a shape.

- 1) G00 - Rapid Motion (without cutting)
- 2) G01 - Straight Line Motion (with cutting)
- 3) G02 and G03 - CW Circular Motion and CCW (with cutting)

NOTE: We are using ZERO'S and not the letter O. The center zero can be left out in the 4 examples above.

All movements that the machine makes in order to machine a workpiece can be divided into one of four types. These commands give you all the power you need to drive the wire through its programmed path.

All of these commands have two things in common. First, they are all modal, meaning that once one of these commands is instated, it need not be programmed in subsequent commands. For example, if you intended to make five straight line motion commands in a row, you only need to program a G01 in the first of the five commands.

Second, each of these commands require that you program the END POINT of the motion desired. Remember that you are telling the machine where you want it to move to in each of these commands. The control will assume that the wire is currently positioned at the beginning point prior to each of these commands.

Look at the following examples.

G00 RAPID MOTION (NO CUTTING)

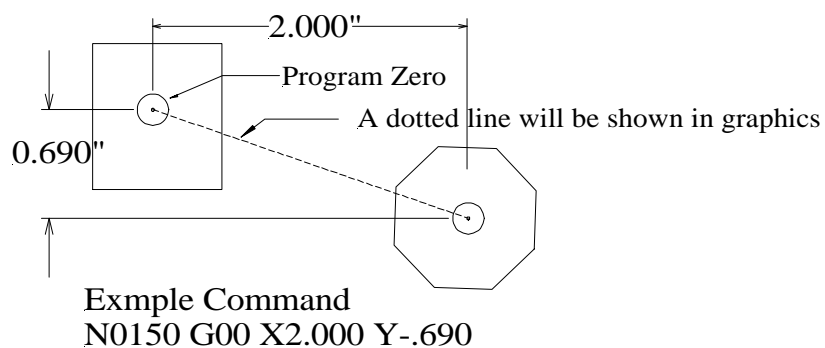
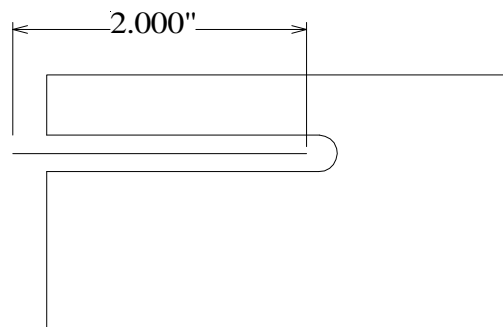


Figure 5-1

The G00 command causes the wire to move at its fastest possible rate. The power supply will not be switched on so no EDMing will be performed. If two (or more) axes are programmed in a G00 command, simultaneous movement of both axis occurs. Be careful with rapid motion commands, as it is possible to crash the machine into a fixture or clamps. NOTE: Rapid moves are not included, when the control calculates the total length of the program.

The primary use for the G00 command (in a program) is to position the wire to a new location. If there are multiple openings in the same part, you can program the machine to rapid to the start hole for the first opening. At this point the wire would be threaded (T91) through the start hole. After the opening is machined, the program would tell the machine to cut (T90) the wire. Then, the next G00 command would tell the machine to rapid to the start hole for the next opening. This could be repeated for as many openings as required.

G01 - STRAIGHT LINE MOTION (ACTUALLY CUTTING)



Example Command
N0150 G01 X 2.000

Figure 5-2

The G01 command is used to cut a straight line. Any number of axes can be included in a G01 command. For example, if you just need to move straight along the X axis, only an X command is necessary in the G01 command. If you need to make an angular motion in X and Y, you can include both an X and Y in the G01 command.

The G01 also turns on the machine functions that allow cutting to occur. Here is what will happen automatically when a G01 is commanded:

- 1) Worktank fills up (when in submerge mode)
- 2) Flushing turns on
- 3) Wire run turns on
- 4) Electricity to the wire turns on using current cutting condition (C value displayed)

As you can see, the G01 command will actually get the machine ready to cut as well as make the straight line cutting motion. Graphics will show solid lines.

G02 AND G03 - CIRCULAR MOTION

G02 is Clockwise motion and G03 is Counterclockwise.

These commands are used to cause the wire to move along a circular path in the cutting mode. They allow you to contour the necessary radii on your workpiece. G02 and G03 also cause the same machine functions to be turned on as G01 such as fill worktank, flushing, wire run and current cutting condition.

SPECIFYING THE X AND Y VALUES IN A CIRCULAR MOVE

In the command line there must be values for the X and Y axis. These values are the X and Y ending point coordinates of the arc.

SPECIFYING THE RADIUS IN A CIRCULAR MOVE WITH I AND J

We must tell the control the radius to be generated by the circular command. We can tell the control this radius value by the I and J in the command line.

I is the **Incremental distance and direction** from the starting point of the arc TO the center of the arc along the X axis.

J is the **Incremental distance and direction** from the starting point of the arc TO the center of the arc along the Y axis.

Depending on the type of move you are trying to make, the I and/or J values may be hard to understand. Here is a trick you can use to help. On your print, draw an arrow FROM the starting point of the arc TO the center of the arc along each axis (the arrow head will always be pointing at the center of the arc). Now ask yourself in which direction each arrow is pointing. If it is pointing in the X minus direction, you will use an I minus value in the circular command. If the arrow is pointing in the Y plus direction, you will use a J plus value in your command (remember that plus is assumed and need not be specified). If the arrow is pointing in both X and Y, you will have both an I and a J in the circular command and the pluses and minuses must also be considered.

The sketches in figure 5-3 and figure 5-4 illustrate this technique:

Pnt / G02 - G03 Example

```

G92X0 Y0
1  G01Y.500
2  G01X.400
3  G02X.500Y.400 J-.100
4  G01Y-.400
5  G02X.400 Y-.500 I-.100
6  G01X-.400
7  G02X-.500 Y-.400 J.100
8  G01Y.400
9  G02X-.400 Y.500 I.100
1  G01 X0
   G01 Y0
   M02

```

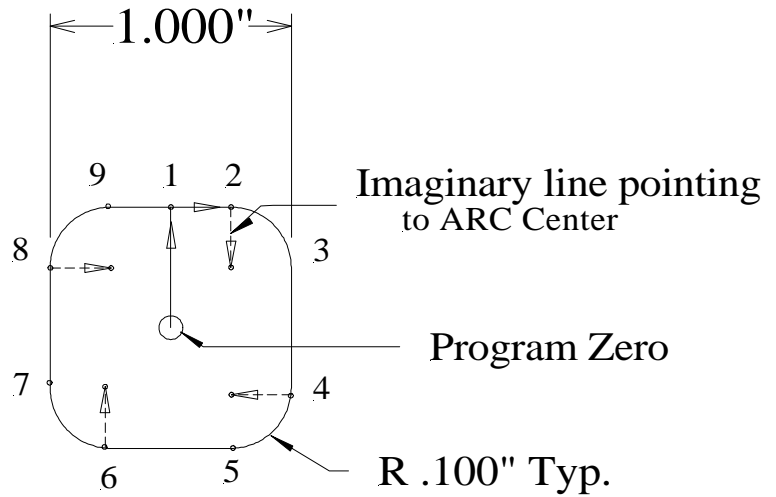


Figure 5-3

Pnt G92 X0. Y0.

```

1  G01Y1.25
2  G02 X.5 I.25 J0.
3  G01 Y.75
4  G03 X1. I.25 J0.
5  G01 Y1.25
6  G02 X1.5 I.25 J0.
7  G01 Y0.
8  X0.
   M02

```

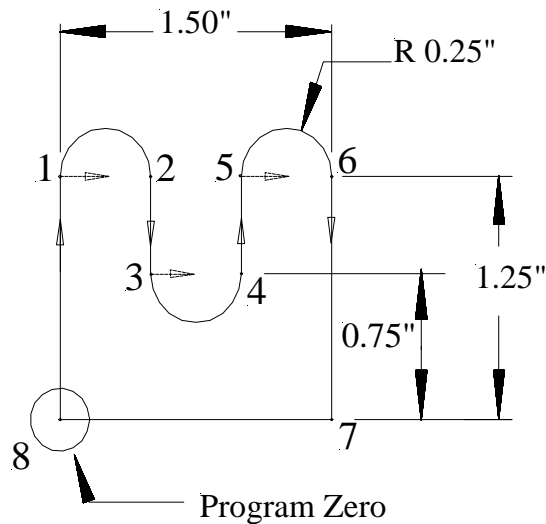


Figure 5-4

CIRCULAR MOTION USING THE R WORD (CONSTANT RADIUS)

Another way to program an arc is by using the R word. To program using the R word, program the SHARP corner of the intersection using G01 then add the R and its value at the end of the command line. Example "G01 X1.234 Y.5678 R.100". This example would cause a .100 radius to be made (think of this as a corner fillet). It should be pointed out that the R word will sound like the easier of the two ways, but when you are machining taper this may not be the type of corner you wanted on the workpiece. This style of programming will work if the program is in absolute G90 or Incremental G91 mode.

Figure 5-5 shows how the R word can be used with a STRAIGHT LINE move (G01) to form a radius when in absolute mode.

Pnt / R word Example

```

      G92X0 Y0
      G42H000
1     G01Y.500
3     G01X.500 R.100
5     G01Y-.500 R.100
7     G01X-.500 R.100
9     G01Y.500 R.100
1     G01 X0
      G40G01 Y0
      M02
  
```

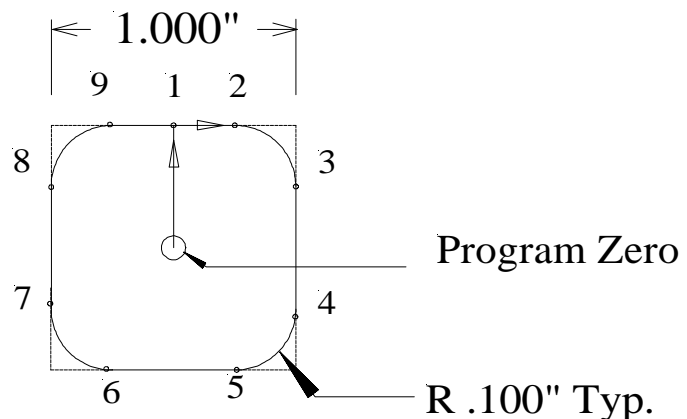


Figure 5-5

Note three things about the R word.

1. G01 is used rather than a G02 or G03 to generate the circular motion.
2. The G01 command tells the wire to move to the intersection point of the next movement (or the sharp corner point). The R word is included to tell the control to start rounding when it reaches the radius value away from the end point. To learn more about the R word, see page 10-14 or look in the code instruction manual which came with your machine.
3. Offset must be used for the R word to work.

TAPER CUTTING CONSIDERATIONS WHEN CUTTING AN ARC:

When taper cutting, if you use the R word to create a circular movement, the control will form the same radius at the top and bottom of the part. If you are machining form tools, the radius generated by an R word is desirable.

This type of radius is usually referred to as an isometric or constant radius.

For die relief, you will want the radius to expand as it reaches the bottom of the die. For this type of radius, use the G02 and G03 style of programming.

For more details, see figure 5-6 or “Automatic Corner Rounding” on page 10-14.

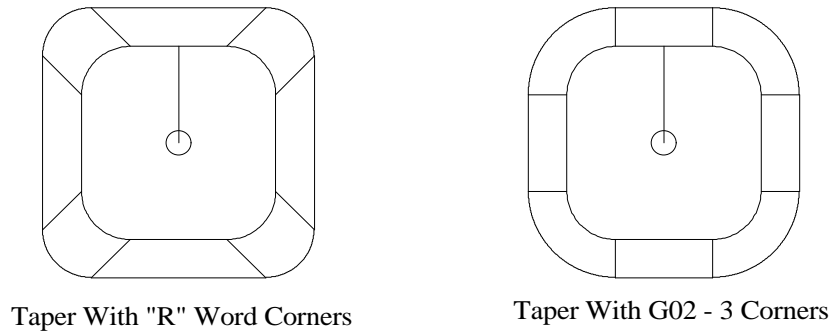


Figure 5-6

EXAMPLE PROGRAM SHOWING THREE MOTION TYPES G00, G01, G03

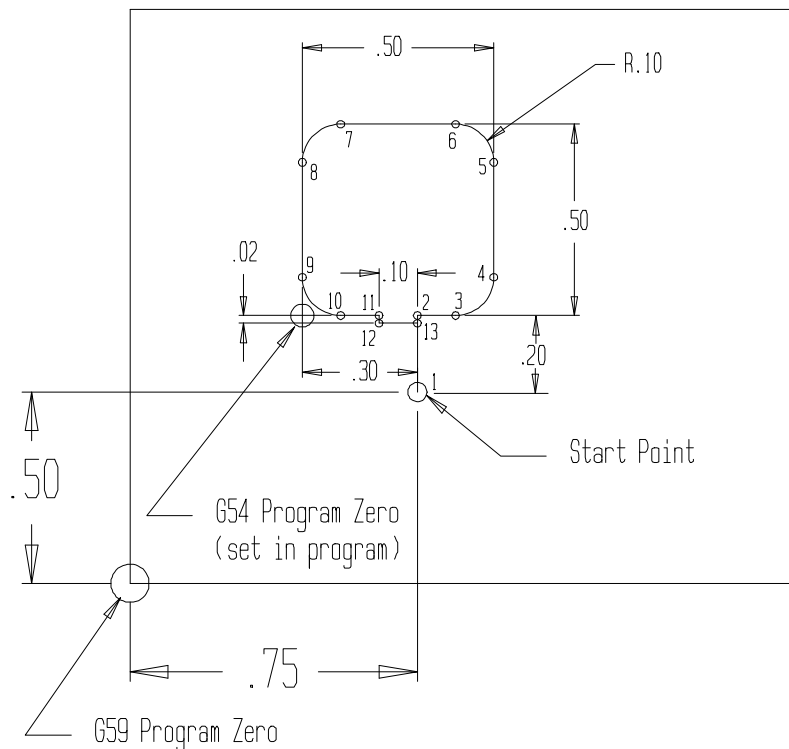


Figure 5-7

(The wire is cut at the start of this program and there is no AWT):
N005 G59 G92X0Y0(VALUE SET FOR G59 DURING SETUP)
N010 G00 X.75 Y.5 (MOVE TO POINT 1)
N015 M00 (PROGRAM STOP TO THREAD WIRE)
N020 G54 (SELECT COORDINATE SYSTEM)
N025 G92 X.3 Y-.2 (SET PROGRAM ZERO)
N030 C411 (SELECT CONDITION)
N035 G90 (SELECT ABSOLUTE MODE)
N037 T84 (TURN ON HIGH FLUSH PUMP)
N040 G42 H062 (SELECT OFFSET DIRECTION AND VALUE)
N045 G01 Y0 (MOVE TO POINT 2)
N050 X.4 (MOVE TO POINT 3)
N055 G03 X.5 Y.1 J.1 (MOVE TO POINT 4)
N060 G01 Y.4 (MOVE TO POINT 5)
N065 G03 X.4 Y.5 I-.1 (MOVE TO POINT 6)
N070 G01 X.1 (MOVE TO POINT 7)
N075 G03 X0 Y.4 J-.1 (MOVE TO POINT 8)
N080 G01 Y.1 (MOVE TO POINT 9)
N085 G03 X.1 Y0 I.1 (MOVE TO POINT 10)
N090 G01 X.2 (MOVE TO POINT 11)
N095 G40 Y-.02 (MOVE TO POINT 12, CANCEL OFFSET)
N100 M00 (STOP TO APPLY MAGNETS OR CLAMP)
N105 G01 X.3 (MOVE TO POINT 13)
N110 Y-.2 (MOVE BACK TO POINT 1)
N115 M02 (END OF PROGRAM)

Careful study of the program will reveal how the part (figure 5-7) is formed using the movement commands just described.

In the following chapters you will learn more of the techniques used in the above program (Offset use).

CHAPTER 6 DESCRIPTION OF PROGRAMMING CODES AND WORDS

This chapter will present most of the codes and words involved with programming. While there will appear to be a great number of them, only a few are used on a regular basis. Try to memorize the codes used most often and what they mean, so you can understand what the program is doing.

/ (slash code)

Used in conjunction with the block skip codes (G11 and G12). When the control reads this slash code, it checks the condition of the block skip function. If block skip is on by the G11 code, the control skips the line of NC code in which the slash code is placed. If block skip is off by G12 code, the control will execute the line of NC code which contains the slash. Skip can also be turned on or off in the SET screen.

Example: G92X0Y0

N0000 G01Y.5000

N0001 X-.5000

N0002 Y-.5000

N0003 X.5000

N0004 Y.5000

N0005 X.1000

/N0006 M00

N0007 X0

N0008 Y0

N0009 M02

In the above example, if the skip code G11 was entered before starting this program, then the machine will not stop at the M00 Code on line 6, as it has the slash code in front of it.

N (sequence number)

Used for numbering the lines in a program. The sequence numbers can be in any order and can even repeat. N is also used for numbering subroutines.

X

Used to specify position movement in the X-axis. A value of 5 inches would be written as X5. or X50000 when in 4 place inch. (Notice where decimal point would be).

Y

Used to specify position movement in the Y-axis. A value of .2 inches would be written as Y.2 or Y2000 when in 4-place inch. (notice where decimal point would be).

Z

Used to specify position movement in the Z-axis when the machine has a power Z-axis. A value of 3 inches would be written as Z3. 5 or Z35000

U

Used to specify exact movements of the upper wire guide (U axis) left and right (parallel to X). Used when writing a program that employs independent UV techniques or 4 axis. U or V 4 axis movements are defined as the incremental value from the X or Y-axis plane in an absolute mode. For standard taper cutting using G51 and G52, the control will calculate the required U and V movements without the need for the U and V codes in the program.

V

The same information applies to V (upper wire guide movement parallel to the Y-axis) as listed above with the U axis.

I

Used in circular movement commands (G02 and G03). I is defined as the incremental distance and direction in X from the starting point of a circular move to the center around which the movement takes place.

J

Used in circular movement commands (G02 and G03). J is defined as the incremental distance and direction in Y from the starting point of a circular move to the center around which the movement takes place.

H

Specifies the register in the offset file to be used for wire radius compensation. A value of .0063 inches would be input as H063 when in 4-digit inch. For 5 digit inch use H630.

A

Used to input the amount of taper angle when using G51 and G52 commands. An angle of 1.0 degree is input as A1.0

P

Used in the line with M98 to designate the sequence number (line number) of the beginning of the sub-routine. When the control executes the line M98 P0010 it jumps to line number N0010 in the subroutine.

L

Can also be used with the sub-routine call to provide multiple executions of the sub-routine (looping). M98 P0030 L4 would execute the sub-routine 30, four times.

C

Designates the three digit cutting condition number which the control will load from the condition file or from the header into the generator. The three digits may be any existing combination from 000 to 999. A Non EF II machine uses 1-199 for the operator to create their own. An EF II machine uses 1-99.

R

Specifies the radius of a corner round. A radius of one inch would be written as R1. or R10000 and a radius of 0.010" would be input as R.01 or R100 (Example NC code G01 X1.0 R.050 this would cause the intersection of the next line to be filleted by .0500)
Note: offset must be turned on for the R code to be used. See page 10-14.

RA

Specifies the amount of rotation about the G92 point of the program. Use G26 to turn it on. Think of the 3 o'clock position as 0 rotation. Rotating a program clockwise is a minus rotation and counter clockwise is a positive rotation. See page 10-16 for more information.

N0000 (PROGRAM START)

N0001 G26 RA 45.0 (rotation angle of 45 degrees)

N0002 G92 X0 Y0 (start point location)

N0003 (rest of program here)

N0004 (rest of program here)

N0025 G27 (cancel rotation)

N0026 M02 (end of program)

Q

Designates a pre-existing routine to be executed. Stored on the user disk or harddrive is a program to pick up the center of a hole. It is stored with the numeric title 0145. To execute this program, type Q0145 in MAL, MDI and press enter and the control will load the program from the user disk or harddrive and execute it.

TP or TN

This is for inputting the taper settings Table To Program and Table To Next in the SET screen. These codes may appear in an NC program to load the taper registers automatically with the value that appears directly after these codes. Used when you want to program the machine to cut taper, holding size at different heights for different openings.

Example:

G92XY

TP2.000 (Sets Table To Program At 2.000)

TN0000 (Sets Table To Next At 0)

G90

LIST OF G CODES

Code	Description	Modal
G00	Rapid Motion (appears as a dotted line in graphics) pg. 5-1	Y
G01	Straight Line Cutting Motion pg. 5-2	Y
G02	Circular Cutting Motion - Clockwise (CW) pg. 5-3	Y
G03	Circular Cutting Motion - Counterclockwise (CCW)	Y
G04	Dwell (ex: G04 x5.0 to dwell for 5 seconds at sharp corner only)	N
G05	X Axis Mirror image (turns CRT readout red when on) pg.10-10	Y
G06	Y Axis Mirror Image (turns CRT readout red when on) pg. 10-10	Y
G08	X-Y Exchange (turns x,y,u,v, to lower case letters when on)	Y
G09	Cancel G05, G06, G08	Y
G11	Block Skip On (activates slash code, /, in NC program) pg. 6-1	Y
G12	Block Skip Off pg. 6-1	Y
G13	AWT REF=1 Wire break recovery with C888 discharge	Y
G14	AWT REF=0 Wire break recovery turned off	Y
G15	AWT REF=2 Wire break recovery without C888 discharge	Y
G20	Inch Mode	Y
G21	Metric Mode	Y
G22	Soft Limit On pg. 15-21	Y
G23	Soft Limit Off pg. 15-21	Y
G26	Pattern Rotation On (ex: G26 RA45.0) pg.10-16	Y
G27	Pattern Rotation Off	Y
G28	Return To G29 Reference Point	N
G29	Reference Point Memory (for auto wire threader) pg. 6-7	N
G30	Return To The Last G92 Location	N
G40	Wire Radius Compensation Cancel pg. 7-1	Y
G41	Wire Radius Compensation Left pg. 7-1	Y
G42	Wire Radius Compensation Right pg. 7-1	Y
G50	Taper Cancel pg. 8-1	Y
G51	Taper Left pg. 8-1	Y
G52	Taper Right pg. 8-1	Y
G54	Work Coordinate System Number 0 pg.3-4	Y
G55	Work Coordinate System Number 1 pg.3-4	Y
G56	Work Coordinate System Number 2 pg.3-4	Y
G57	Work Coordinate System Number 3 pg.3-4	Y
G58	Work Coordinate System Number 4 pg.3-4	Y
G59	Work Coordinate System Number 5 pg.3-4	Y
G74	4 Axis On	Y
G75	4 Axis Off	Y
G80	Sensor touch pg.14-5	Y
G81	Move To Limits pg.14-1	N

Code	Description	Modal
G82	Move To Half Of Axis Display pg. 10-7	N
G83	Read Current Position and Store To An H Register pg. 10-7	N
G90	Set Absolute Mode	Y
G91	Set Incremental Mode	Y
G92	Assign Values to Axis Displays In Current Work Coordinate p3-3	N
G97	Assign Values to Axis Displays In ALL Work Coordinates	N
G104	Dwell At All Sharp Corners (ex: G104 X5.0)	Y
G105	Cancel G104 code	Y
G140	4Axis Offset Off	N
G141	4Axis Offset Left	Y
G142	4Axis Offset Right	Y
G154 TO G159	Work Coordinate Screen 10 to 15 pg. 3-4	Y
G254 TO G259	Work Coordinate Screen 20 to 25 pg. 3-4	Y
G354 TO G359	Work Coordinate Screen 30 to 35 pg. 3-4	Y
G454 TO G459	Work Coordinate Screen 40 to 45 pg. 3-4	Y
G554 TO G559	Work Coordinate Screen 50 to 55 pg. 3-4	Y
G654 TO G659	Work Coordinate Screen 60 to 65 pg. 3-4	Y
G754 TO G759	Work Coordinate Screen 70 to 75 pg. 3-4	Y
G854 TO G859	Work Coordinate Screen 80 to 85 pg. 3-4	Y
G954 TO G959	Work Coordinate Screen 90 to 95 pg. 3-4	Y
FLAG2D0	M01 code off pg. 15-14	Y
FLAG2D1	M01 code on pg. 15-14	Y
FLAG3D****	scale factor input pg. 10-13	Y

LIST OF T CODES

Code	Description
T80	Wire Drive On
T81	Wire Drive Off
T82	Worktank Drain Closed
T83	Worktank Drain Open
T84	High Pressure Flushing Mode On (high flush)
T85	High Pressure Flushing Mode Off (low flush)
T86	Flushing Pump On
T87	Flushing Pump Off
T89	Switch To Non-Submersed Machining
T90	Cut Wire By Auto Wire Threader Sequence One
T91	Thread Wire By Auto Wire Threader Sequence Two
T94	Switch To Submersed Machining
T96	Worktank Fill Pump On
T97	Worktank Fill Pump Off

Note: Out of the above T codes, the only codes that are usually used in an NC program

would be T84 - T85 to turn on or off high flushing, and T90 -T91 to cut and thread the wire

The other T codes may be used if so desired, but the black switches on the control normally activate them.

LIST OF M CODES

Code	Description
M00	Program Stop
M01	Optional Stop
M02	End Of Program
M03	AWT Jump
M05	Sensor Touch Off pg. 10-7
M06	Disable EDM Power During Movement.
M33	Rotate Wire Tip Disposal Arm Forward. Use to remove the wire tip container.
M43	Rotate Wire Tip Disposal Arm Back.
M98	Jump to a Sub-Routine from the main program.
M99	Return from a Sub-Routine to the main program.

M CODE DESCRIPTIONS

M00 Program Stop (If located at the end of the NC line it may not appear in graphics.

M01 Optional Stop (This Parameter in the SET screen must be set to ON to enable recognition of M01 or by using FLAG2D1 in the program).

M03 AWT Jump : If the T91 operation fails (thread wire) at the present position, then the program will be skipped up to the next M03 position. The M03 parameter in the SET screen must be set to 1, for this code to be active. To use this function in your programs, simply add the M03 code before the T90 codes.

Example:

G92X0 Y0

G29

M98P0000

M03 (This first M03 here is not really needed, but is OK)

T90

G00X1.500 Y2.500

T91 → If wire tread fails, then skip to next M03 code

M98P0001

M03 ←

T90

G00X2.500 Y2.500

T91

M98P0002
M02

ADDITIONAL INFORMATION FOR POST PROCESSOR CREATION

DISK FORMAT: The floppy drive on the Sodick EX21 and Mark 21 control is capable of using 720KB type 3.5-inch floppies – “2DD” type. Disks formatted for IBM use will be fine, just be sure to format at 720Kb (not 1.44Mb). If the floppy is a “2HD” type ***TAPE CLOSED THE SQUARE KNOCKOUT ON THE LOWER RIGHT OF THE DISK!***

DECIMAL POINTS: The above examples use decimal points in the program, this is not required but is recommended.

OFFSET VALUE OUTPUT FORMAT: The above examples are assuming the machine is in 4-digit inch. Please follow the below table. The machine can be placed in 5 digit if preferred.

<u>4 Digit inch offsets</u>	<u>5 Digit inch offsets</u>	<u>Actual amount</u>
H089	H890	.00890
H052	H520	.00520

SPECIAL CODES FOR AWT (Threader equipped)

Cut - Thread wire example

N0100 M03
N0101 T90
N0102 G00 X (value) Y (value)
N0103 T91

G29 code: Use this code to set the present position as wire rethread point (if the wire breaks). Usually used at the beginning of a program if the T91 code is not there. Also use this code before tab cut off moves when making punches, as the T91 code probably was not needed to thread the wire, but a roughing condition will be used for the tab move.

N0001 G54
N0002 G90 G92 X0 Y0
N0003 G29
N0004 T84
Etc.

G02 G03 “I” or “J” Values: I or J values are incremental from start of arc to the center point of the arc.

G74 (4 axis) PROGRAMMING “U” or “V” Values: U or V axis values are incremental from the X Y values, in absolute from it's present position.

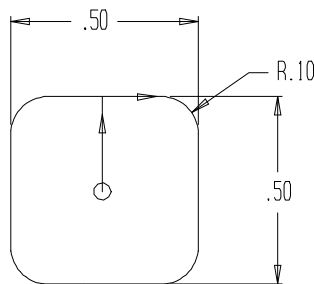
DECIMAL POINTS: The control will accept decimal points, and is preferred, but not mandatory.

TRAILING ZERO'S: Trailing zero's after the output value, when using decimal points, are not required. EX “G02 X.1 Y.12 I.123 J.1234”.

NC PROGRAM EXAMPLES

PROGRAM EXAMPLE 1

The below program is an example to make a .500” square, with .100” corners. The approach is at a right angle to the contour, and starts at X0 Y0.



```

N0001;
N0002 G54 G90
N0003 G92 X0. Y0. G29
N0004 T84 (High Flush)
N0005 T94 (Submerged)
N0006 C451 (Cutting Condition)
N0007 G42 H082 (Offset Right, .0082")
CRT(JOB 1 CNT2X ROUGH CUT FORWARD)
N0008 G52 A.5000 G01 Y.25 (Taper Right, .500 Degree)
N0009 X.15
N0010 G02 X.25 Y.15 I0. J-.1
N0011 G01 Y-.15
N0012 G02 X.15 Y-.25 I-.1 J0.
N0013 G01 X-.15
N0014 G02 X-.25 Y-.15 I0. J.1
N0015 G01 Y.15
N0016 G02 X-.15 Y.25 I.1 J0.
N0017 G01 X-.1
N0018 X0.
N0019 G50 A0. G40 Y0. (Cancel Offset and Taper)

```


CRT(JOB 1 CNT2X SKIM 1 CUT REVERSE)

N0020 T85 (Low Flush)

N0021 C623

N0022 G41 H053

N0023 G51 A.5000 G01 Y.25

N0024 X-.15

N0025 G03 X-.25 Y.15 I0. J-.1

N0026 G01 Y-.15

N0027 G03 X-.15 Y-.25 I.1 J0.

N0028 G01 X.15

N0029 G03 X.25 Y-.15 I0. J.1

N0030 G01 Y.15

N0031 G03 X.15 Y.25 I-.1 J0.

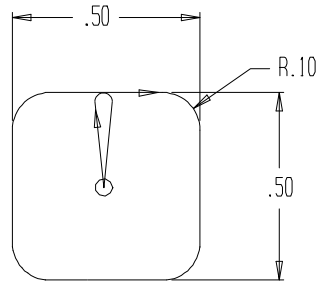
N0032 G01 X0.

N0033 G50 A0. G40 Y0.

N0034 M02

PROGRAM EXAMPLE 2

The below program is for the same shape as prior, but uses a blend on radius of .1000”.



N0001

N0002 G54 G90

N0003 G92 X0. Y0. G29

N0004 T84

N0005 T94

N0006 C451

N0007 G42 H082

CRT(JOB 1 CNT2X ROUGH CUT FORWARD)

N0008 G52 A.5000 G01 X-.1 Y.15

N0009 G02 X0. Y.25 I.1 J0.

N0010 G01 X.15

N0011 G02 X.25 Y.15 I0. J-.1

N0012 G01 Y-.15

N0013 G02 X.15 Y-.25 I-.1 J0.

N0014 G01 X-.15

N0015 G02 X-.25 Y-.15 I0. J.1

N0016 G01 Y.15

N0017 G02 X-.15 Y.25 I.1 J0.
N0018 G01 X-.1
N0019 X0.
N0020 G02 X.1 Y.15 I0. J-.1
N0021 G50 A0. G01 G40 X0. Y0.
CRT(JOB 1 CNT2X SKIM 1 CUT REVERSE)
N0022 T85
N0023 C623
N0024 G41 H053
N0025 G51 A.5000 G01 X.1 Y.15
N0026 G03 X0. Y.25 I-.1 J0.
N0027 G01 X-.15
N0028 G03 X-.25 Y.15 I0. J-.1
N0029 G01 Y-.15
N0030 G03 X-.15 Y-.25 I.1 J0.
N0031 G01 X.15
N0032 G03 X.25 Y-.15 I0. J.1
N0033 G01 Y.15
N0034 G03 X.15 Y.25 I-.1 J0.
N0035 G01 X0.
N0036 G03 X-.1 Y.15 I0. J-.1
N0037 G50 A0. G01 G40 X0. Y0.
N0038 M02

CHAPTER 7 WIRE OFFSET DESCRIPTION

UNDERSTANDING OFFSETS

An “offset” tells the control how far to keep the wire centerline away from the programmed surface to be cut for any given pass. If making only one pass to size (roughing the part right to size), the offset would be equal to the wire radius + the amount of the spark gap. If making multiple passes, the amount of offset used per pass will be slightly larger than the wire radius + spark gap. It must be large enough to leave stock on the part for the next pass.

There are three “G” codes related to wire radius compensation:

G40 - Cancel

G41 - Wire Left

G42 - Wire Right

THREE ADVANTAGES TO USING OFFSETS

The first advantage of this feature allows the operator to input actual part coordinates into the program with no concern for the wire diameter in the values of the coordinates. The control will automatically keep the wire a specified distance (the offset value) away from the surfaces of the part while cutting. This generates the movements necessary to make the part to the proper size.

The second advantage to using wire offsets is that the same program that machined the first rough pass can be used for subsequent trim passes. This means you may not have to develop multiple programs for the various trim passes required to make the part to the size and finish requirements you need.

The third advantage to using wire offset is that you will be able to incorporate die clearance into your program with the same set of coordinates that machined the part to size.

OFFSET PROGRAMMING

Now that you know some of the advantages to using wire offsets, let's look at how you use it. Wire offsetting requires three steps in the program:

- 1) Instate wire offset (G41 or G42 codes)
- 2) Drive the wire through its motions
- 3) When finished, you MUST cancel offset (G40 code)

INITIALIZING OFFSET

First, let's look at how you instate offset.

To instate offset, you will be choosing from G41 (wire left) and G42 (wire right). You must be able to decide how the wire will be related to the workpiece during the cutting motions. To evaluate this relationship, look in the direction that the wire will be moving during the cut (rotate the print, if necessary). Looking in this direction, ask yourself, "What side of the programmed path is the WIRE on? Is the wire on the left side or the right side of the path?"

If the wire is on the left side of the path, you will use G41 to instate wire radius offset. If the wire is on the right side of the path, you will use G42.

TIP: It is wise to make yourself a general rule for radius compensation. If you make the arbitrary decision that you will ALWAYS machine workpieces in a clockwise direction around the part, dies (or any inside shape) will ALWAYS use G42 and punches will ALWAYS use G41. Of course, if you decide to machine workpieces in a counter clockwise direction around the part the rule will change. Also note that if you are making reverse direction trim passes, the offset will change for each trim pass (G41 for the first pass will change to G42 for the next reverse direction pass). Figures 7-1A and 7-1B should illustrate this concept.

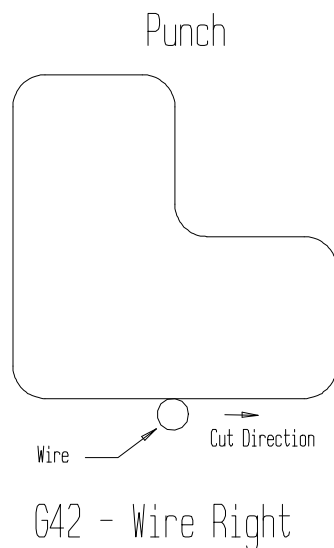


Figure 7-1A

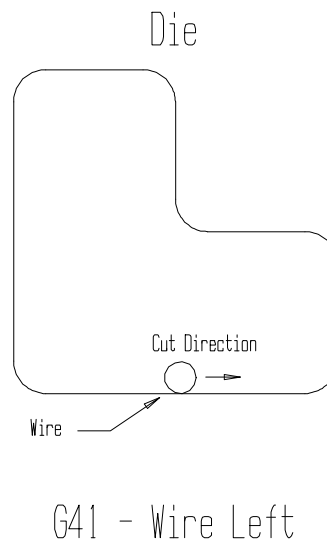


Figure 7-1B

Now that you can decide between G41 and G42, let's see how you initialize a wire compensation value. It is important to use offset PRIOR to the first cutting move so that as you begin cutting, the first piece of geometry to be cut will be on size. As the wire moves to the end point of that move, it will automatically be offset so that the edge (plus the amount of overburn) of the wire is flush with the surface you wish to be on size.

OFFSET VALUE

In the initialization block, you will include an “H” word. This specifies the register the control will use in the offset file. The H word tells the control how far to keep the wire away from all surfaces to be cut. For example, if you needed an offset of .0062", it would be programmed with the H word as H062. Use the following table as a guide, based on the DIGIT parameter setting in the “SET” “TRAVEL” screen.

<u>4 Digit inch offsets</u>	<u>5 Digit inch offsets</u>	<u>Actual amount</u>
H089	H890	.00890
H052	H520	.00520

Driving the wire around the shape

Now that wire radius compensation is properly instated, you will simply program the part coordinates to drive the wire through its programmed path to cut the workpiece.

CANCELING WIRE RADIUS COMPENSATION

When you are finished with your cutting movements, you **MUST** remember to cancel the wire radius compensation (with G40). It is important that you do so in a straight line cutting command (not G02 or G03). At the end of the cancellation command, the wire's centerline will be on the programmed end point of the command.

“APPROACH PATTERN” PARAMETER

The parameter “Approach Pattern” in the “SET” “TRAVEL” screen affects the way in which the offset is instated on the first move and how it's cancelled on the last move. It is recommended that the parameter be set to 1. The figure 7-2 shows the affect it has on these moves. When “Approach Pattern” is set to 1, the wire approaches at a **right angle** to the surface to be offset from. If the parameter is set to 0, then the offset value is applied and cancelled along the length of the initialization move.

"Approach Pattern" And Wire Offset Initialization

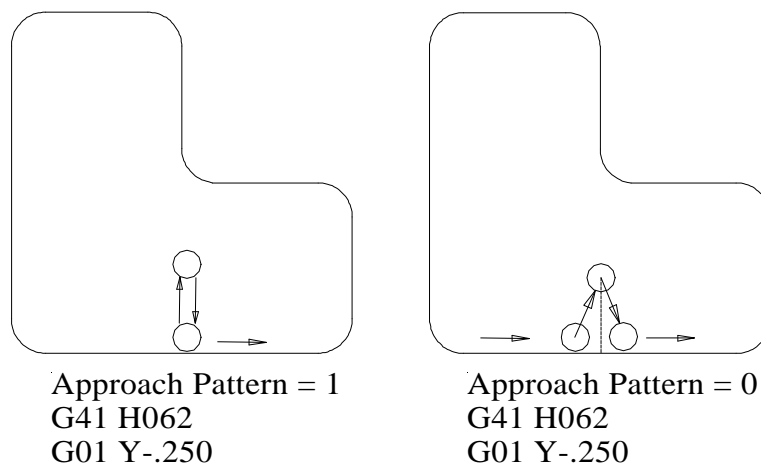


Figure 7-2

Now let's put it all together. Figure 7-3 is the drawing for a sample program that shows how to program wire radius compensation correctly. Note the method by which the compensation is instated and canceled. This example is showing the method of using wire radius compensation and makes no trim passes.

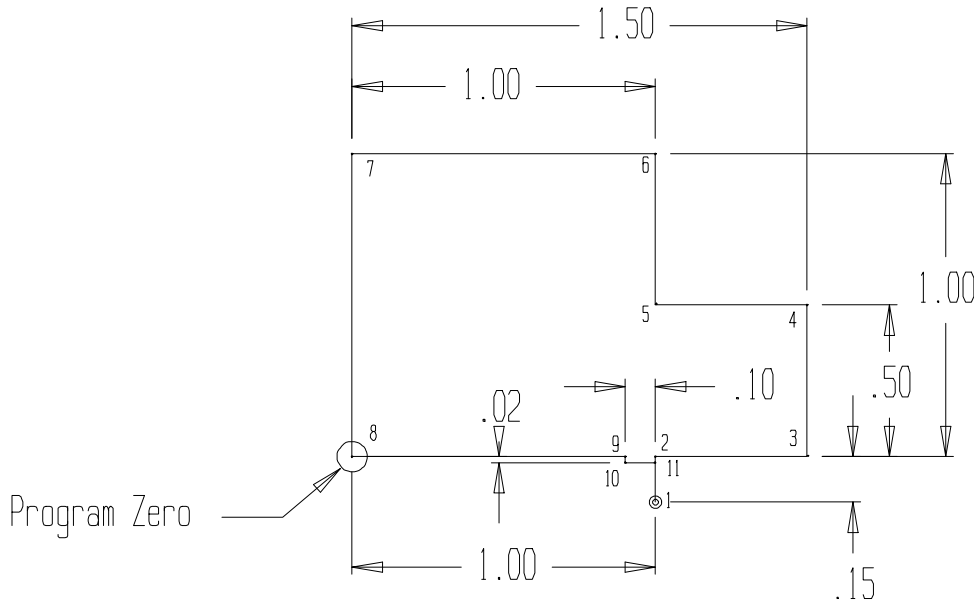


Figure 7-3

Program:

```

N005 G54 (SELECT COORDINATE SYSTEM NUMBER ZERO)
N010 G92 X1. Y-.15 (SET PROGRAM ZERO)
N015 G90 (SELECT ABSOLUTE MODE)
N020 G42 H061 (SELECT WIRE RIGHT AND .0061 OFFSET VALUE)
N025 C411 (SELECT MACHINING CONDITION)
N026 T91 ( AUTO WIRE THREAD)
N030 G01 Y0 (MOVE TO POINT 2)
N035 X1.5 (MOVE TO POINT 3)
N040 Y.5 (MOVE TO POINT 4)
N045 X1. (MOVE TO POINT 5)
N050 Y1. (MOVE TO POINT 6)
N055 X0 (MOVE TO POINT 7)
N060 Y0 (MOVE TO POINT 8)
N065 X.9 (MOVE TO POINT 9)
N070 G40 Y-.02 (MOVE TO POINT 10 AND CANCEL COMP.)
N075 M00 (PROGRAM STOP TO APPLY MAGNETS)
N080 G01 X1. (MOVE TO POINT 11 TO CUT OFF)
N085 Y-.15 (MOVE BACK TO START POINT)
N090 M02 (END OF PROGRAM)

```

IMPORTANCE OF TEST SQUARES

If you are about to cut a part and are unsure about something, it would be FOOLISH to cut before you test the conditions and offsets to prove that they are correct. For this reason, we recommend that any time you are unsure about something, you cut a test square.

A test square is made from the same material and thickness as the workpiece to be machined. And, of course the same wire diameter will be used to machine the test square as the actual part. It is preferable to actually use the same piece of material that is going to be machined as the material for the test square. If machining a die, choose an open area of the die in which to run the test square. If the part is to be a punch, find another area of the stock to run the test square so that it will not interfere with the machining of the punch. In the planning stages, when drilling the start hole for the punch or die, be sure to plan for a start hole for the test square!

Now, write a program to machine the test square (usually .500 inch square) that will use the same conditions and offsets to be used to machine the punch or die. After you machine the test square, you can see the finish and measure the size that the set of conditions and offsets will generate. If the test square is on size, so will any punch or die machined with the set of conditions and offsets for this material, thickness, and wire diameter.

While machining the test square, do not change any of the parameters related to the condition. If you make changes, another test square must be cut, since you are changing values that determine part size, accuracy, and finish.

TEST CUTTING TO CONFIRM OFFSETS

If you find that the finish and size of the part is unacceptable, the set of conditions and offsets must be modified. For beginners, it would be wise to contact Sodick for help. If the finish is OK but the size of the part is incorrect, you will have to modify ALL offsets by the amount of error. Here's an example. Say the test punch was programmed to be .500 square, but it came out to .4996 square. The test square is .0004 (overall) too small. In this case, ALL offsets would have to be INCREASED by .0002 inch to correct the error.

Once a set of conditions and offsets has been proven to give the desired results, they will continue to be correct. This means that once you know that a set of conditions and offsets are cutting properly, you can rest assured that they will remain good for the material, part thickness, and wire diameter for as long as you run the machine.

With a brand new machine, it may seem tedious to machine test squares. However, depending on the accuracy, finish you expect and operator knowledge, they are

necessary. It may seem like a great deal of work to cut test squares for each material, thickness, and wire diameter you are going to be using, but it is the only way to know what the results will be.

INCORPORATING DIE CLEARANCE WITH OFFSETS

It is important to cut test squares to attain the exact values for your on size offsets. The on size offset if used will make the part come out to the EXACT size that the part coordinates in the program commanded. As you also know, you can lie to the control about the values of the offsets to force it to machine the part with clearance.

Once you have determined the amount of clearance that is necessary, you can easily use wire offsets to create the clearance. To come up with the modified offsets for the shape that needs clearance, simply SUBTRACT the amount of clearance (per side) from ALL offsets needed to make the shape on size. By subtracting this amount from all of the on size offsets, you will be making punches smaller and dies bigger. Here is a detailed description.

Say you had a punch and die to make and you are going to use .010 diameter wire. You determine that the punch must be made on size and the die must incorporate .002" die clearance (per side). You look in the condition manual and have decided that you need to use three cuts (rough and two trims) to make the part to your finish and accuracy requirements. Lets assume the book recommends the following offset values.

Rough Pass .0085 inch (H085).

Second pass .0056 inch (H056)

Third pass .0053 inch (H053).

You test the recommended conditions and offsets on a test square and find the results acceptable.

You would cut the punch with the recommended offsets (H085, H056, and H053). When you machine the die, you will take the on size offsets and SUBTRACT .002" from all the offsets to come up with the offsets for a .002 inch die clearance.

The die offsets with the clearance calculated in would be as follows.

Rough Pass .0065 inch (H065).

Second pass .0036 inch (H036)

Third pass .0033 inch (H033).

These offsets will keep the wire .002 **closer** to the programmed path than the on size offsets, and correctly form the die clearance.

WHEN CLEARANCE IS LARGER THAN THE OFFSET VALUE

If the clearance is larger than the smallest on size offset, when you subtract the die clearance from the on size offset, the result will be a **negative H number**. In this case the H number can be called up as a negative value EX: G41 H-050 (negative .005 inch

value). This negative call out is new for Sodick, starting with the Mark21 and EX21 controls. But this is a good time to point out that there is another way of setting offsets (positive or negative) using what we call a HEADER. Cutting conditions can also be entered into the header using C registers.

USING A “HEADER” IN THE NC FILE

A “Header” is the portion at the top of your NC file, which can contain the cutting condition and “H” registers for the current program to use.

There are six H-registers at the header of any program input into “New File” ranging from H000 to H005. Actually the values of the H registers do not matter and you can change them if you prefer, but your NC program must call these H registers to use them. Below is a sample of these H registers from newfile. NOTE: Decimal points cannot be used in the header.

(NEWFILE)

H000 = +00000000 H001 = +00000000 H002 = +00000000
H003 = +00000000 H004 = +00000000 H005 = +00000000

When you specify H001 in the program after the G41 or G42 codes, the control will look for H001 register in the header of the program and use the value that is stored in that H register. If there is not an H001 register in the header, it looks in the *offset file* for register H001 and uses the value in that register (default value of H001 is .0001”).

ENTERING VALUES FOR THE H CODES OF NEWFILE

You can enter either plus or minus values in the header H registers. Lets say you needed .010" clearance (per side) on a part that is going to be cut.

You could use H001 for the first pass, H002 for the second pass, and H003 for the third pass. But before running the program, insert the values of the offsets into the header H registers. Here is what the header would look like prior to running the program with .010 die clearance:

NC Program Example: *H1 – H2 – H3 offset data is input manually in EDIT screen*

H000 = +00000000 H001 = -00000015 H002 = -00000044
H003 = -00000047 H004 = +00000000 H005 = +00000000

G92X0Y0

T84

C1

G41H1 (*This calls the value of H1 which was set into the Header above*)

G01X.5

Y.5

X0

G40Y0

T85

C2 *(CONTINUES ON NEXT PAGE)*

G41H2 *(This calls the value of H2)*

G01X.5

Y.5

X0

G40Y0

T85

C3

G41H3 *(This calls the value of H3)*

G01X.5

Y.5

X0

G40Y0

M02

Notice that the offset values set into the Header were calculated as follows.

Book Offset - Clearance / side = Header value

.0085" - .010" = -.0015"

.0056" - .010" = -.0044"

.0053" - .010" = -.0047"

Because H000, H004 and H005 are not needed in the above example, (the NC program doesn't call out these registers) you could delete them from the header. Another header example follows, to get the same result.

H001 = -00000015 H002 = -00000044 H003 = -00000047

SETTING NEGATIVE OFFSET VALUES DIRECTLY

If your program does not use a Header then you can call out negative offset values from the *offset file* directly. Below is an example of how to do this.

NC Program Example:

G92X0Y0

T84

C1

G41H-015 *(This calls out the value of H015 and switches it negative)*

G01X.5

Y.5

X0

G40Y0

T85

C2

G41H-044 *(This calls out the value of H044 and switches it negative)*

G01X.5

Y.5

X0
G40Y0
T85
C3
G41H-047 *(This calls out the value of H047 and switches it negative)*
G01X.5
Y.5
X0
G40Y0
M02

ALTERNATE METHOD OF DEFINING OFFSET VALUES

Another way to set offset values is to define a register content, then call out that register with the G41/G42 codes. This method is for advanced programmers, so if it seems confusing to you, use the method described above. Below is an example.

NC Program Example:

G92X0Y0
T84
C1
H1=-.0015 *(Sets H1 register content to -.0015")*
G41H1 *(Calls the H1 register, which was set to -.0015" above)*
G01X.5
Y.5
X0
G40Y0
T85
C2
H1=-.0044 *(Sets H1 register content to -.0044")*
G41H1 *(Calls the H1 register, which was set to -.0044" above)*
G01X.5
Y.5
X0
G40Y0
T85
C3
H1=-.0047 *(Sets H1 register content to -.0047")*
G41H1 *(Calls the H1 register, which was set to -.0047" above)*
G01X.5
Y.5
X0
G40Y0
M02

This Page Left Blank

CHAPTER 8 TAPER CUTTING INFORMATION

Now we will discuss taper cutting. We will limit our discussions about taper cutting to machining die clearance angles. There are other applications for taper cutting, but we feel that if you understand how to machine taper for die clearance angles, you should be able to apply what you know to other applications.

The primary use for machining taper is for die clearance. You usually need some form of clearance angle on the die to allow the blanked slugs from the punched workpiece to fall easily through the die. There are several different ways to machine this taper, depending on your requirements for cycle time, clearance angle, and basic personal preferences.

TAPER CUTTING CONSIDERATIONS:

Do you want Die Land? Die land (also called die life) is an amount of non-tapered area of the die. The purpose for die land is to allow the die to be sharpened (ground on the top surface) with absolutely no deviation to the size of the die opening. Depending on the taper angle, die land may not be necessary. Many companies machine only about .25 degrees (1/4 degree) taper as the angle. If your desired taper angle is this small, and if the part to be punched is relatively thick, the very small change in die opening size during sharpening will be minimal. For example, if machining .25 degrees as the taper angle, when you sharpen the die and remove .007 inch stock from the top surface of the die, the change in die opening size will only be .0000305 inch (per side). For all intents and purposes, this small deviation will not affect the performance of the die.

Knowing this, companies can save a great deal of cycle time by tapering right up to the top of the die with absolutely no land. But remember, as the taper angle grows, the amount of deviation to the die opening during sharpening will also grow. You can easily calculate the amount of growth (per side) by applying this formula:

Growth (per side)= TAN of the taper angle times the amount of stock to be removed per sharpening:

How much die land do you need? Of course, the amount of die land will be directly related to the life of the die. But you must also be concerned with the ease of the blanked part removal. You can reference the Machinery's Handbook to get their recommendation for amount of die land for various blanked part thickness.

How big do you want the taper angle to be? The taper angle in a die affects the strength of the die. Generally, you will want the taper angle to be at its smallest acceptable angle to keep the die strong. The desired taper angle will also affect the best and easiest way to machine the die. More on this later.

Maximum Taper Angle

The maximum taper angle *per side* can be calculated for a particular workpiece thickness. To determine the maximum taper angle that can be cut at a given thickness, use the following calculation (*angle must never exceed 30°*).

$$\text{Max Angle} = \text{TAN}^{-1} [\text{U axis travel per side} / [\text{part thickness} + .800]]$$

The models A320D or A530D have .980" of U axis travel per side.

The models AP350 or AP500 have 1.380" of U axis travel per side.

Do you want to cut the die with the top of the die up or down? For most applications, it is best to cut the die with the top of the die up. It makes it easier to measure the size of the die while it is still on the table of the machine. However, you must also be concerned with getting the slug out after the first roughing pass! If the wire diameter is very small, and the taper angle is very large, it may be impossible (for large dies) to remove the slug from the top after the first roughing passes. If you multiply the tangent of the taper angle times thickness of the part you are machining, you can easily tell if the slug can be machined from the top of the die. If the value you calculate is **SMALLER** than the wire diameter you are using, you will be able to remove the slug from the top. If not, you might want to turn the die over and machine with the top of the die face down on the table top (for large dies), rather than face the necessity of moving X and/or Y to remove it from the bottom. Note: removing it from the top would be mandatory if the die filled the work area.

Now that you know some of the considerations to be concerned with, let's look at some recommended methods of machining dies with taper.

TAPERING TO THE TOP OF THE DIE (NO DIE LAND)

This is the easiest way to machine a die. With this method, you only need one program that will rough and finish the entire die. When programming, you specify the taper direction (left or right) and the taper angle correctly. Of course, you must also give consideration to your offsets, based on whether the punch or the die requires the clearance (to allow the punch to fit properly into the die).

MACHINING A DIE WITH A LAND

There is a question as to whether it is smarter to machine the die land first and then the taper angle, or whether it is smarter to machine the taper angle first and then the land. Much of this is based on personal preference. Depending on the taper angle and thickness of the die land, we recommend that you run the taper angle first and then machine the land. However, this may not be possible if the taper angle or die land thickness is too large.

We recommend this because the total cycle time to machine the die would be dramatically reduced with this method. If you machine the die land first, the entire shape

being machined must be finished to the proper size in the (straight) land area of the die. Then the taper angle must be machined. If several trim passes are necessary, this could almost double the production time for the die. By using our recommended method, the taper angle is cut first (with roughing conditions), leaving only a very small area to be machined for the land area of the die. The necessary trim passes may be reduced, but even if not, the surface area for machining this area is reduced to the point that the trim passes will occur much faster.

Say for example that you are machining a .25 degree taper angle and the thickness of the die land is to be .125 inch. In this case, if the clearance angle is machined first, the amount of stock left for the land to be machined is only .00054 inch, and one trim pass would easily remove this stock. To calculate the amount of stock for the die land area simply multiply the tangent of the clearance angle times the thickness of the land.

However if the taper angle were 5 degrees and the land thickness were .25 inch, the amount of stock would be .0218", which is greater than the wire diameter. In this case, if you tried to use our recommended method the wire would actually be creating a slug as you machined the die land. This slug would cause problems during cutting and in this case you would be wiser to machine the die land first.

If the stock left in the die land area after machining the clearance angle is greater than about half the wire diameter, then you must cut the die land first. If not, you can save a great deal of time by cutting the taper angle first.

There is one exception to this rule. If you are machining within extremely accurate tolerances (.0001 or better), you may be wiser to machine the die land first in all cases. But for general-purpose applications, use our general rule.

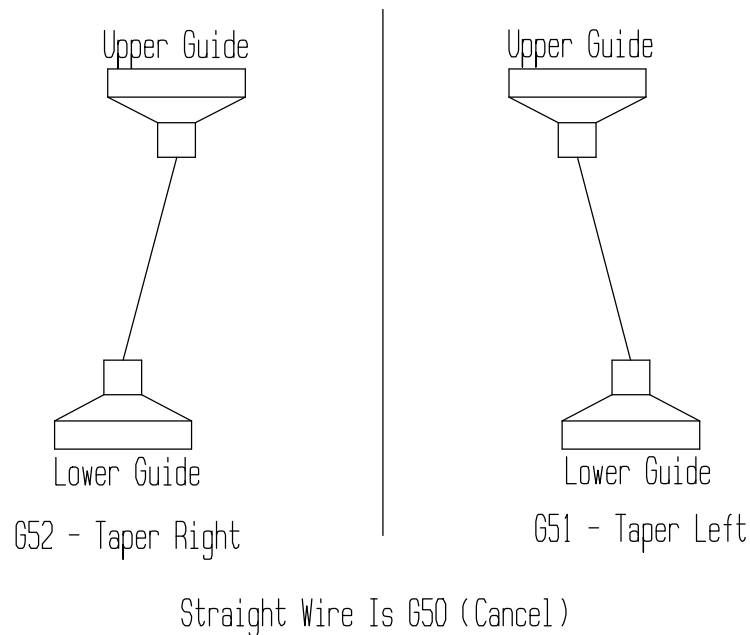
HOW TO PROGRAM TAPER

Now that you know the theory of taper cutting, you can decide for yourself which method of taper cutting is best for your particular application. We will give you the information to do it any way you wish.

As with wire radius compensation, there are three steps to programming taper:

- 1) Initialize taper (G51 for left, G52 for right)
- 2) Make machining motions
- 3) Cancel taper cutting mode (with G50)

(Notice the similarities to wire radius compensation. These commands should be easy to remember!)

*Figure 8-1*

INITIALIZING TAPER

First let's discuss initializing taper correctly. To initialize the taper cutting mode, you must look in the direction the wire will be moving during the cut (rotate print if necessary). Looking in this direction, ask yourself, "Which way does the UPPER GUIDE have to move to generate the desired taper, left or right?" If the UPPER GUIDE has to move to the left, you will use G51 to initialize taper. If the UPPER GUIDE has to move to the right, you will use G52 to initialize taper.

As with wire radius compensation, it is wise to set yourself a general rule to help you remember which direction you need. If you always cut dies with the top of the die UP, you will use G51 (left) whenever making counterclockwise (general direction) passes around the die and G52 (right) when making clockwise direction passes around the die. Of course, if you cut with the top of the die down, the rule will change.

Figure 8-1 illustrates this.

Now that you can decide between right and left taper (G51 or G52), let's discuss the ANGLE you must also include in the initialization command. You will be using an "A" to command the desired taper angle. If you want to machine a half-degree taper (per side), the program needs A.5 in it. You also need to know that there is a limitation of the

maximum possible taper angle on your particular machine based on part thickness. The thinner the workpiece, the greater the taper angle can be.

To initialize taper properly, you simply program the G51 (left) or G52 (right) in a command with the “A” word. Then move to the first surface you wish to machine with taper. The control will initiate the taper DURING the movement to the first surface to cut.

Once initialized correctly, the taper cutting commands will cause the wire to maintain the proper angle around all coordinates programmed.

After all the cutting movements have been made, you **MUST** remember to cancel taper cutting (with a G50 command). The control will bring the wire back to vertical during the next X and or Y movement.

EXAMPLE OF HOW TO PROGRAM TAPER CORRECTLY:

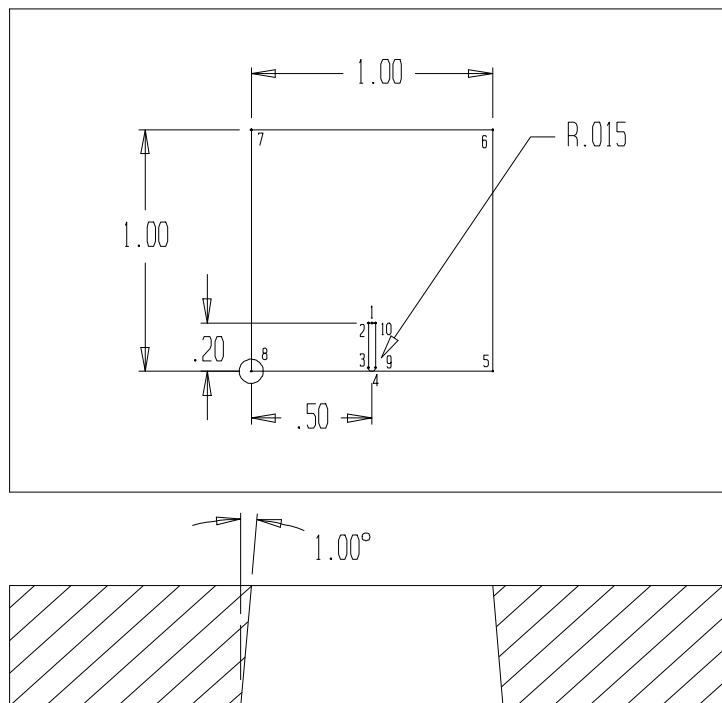


Figure 8-2

NC program for figure 8-2

```
N005 G54 (SELECT COORDINATE SYSTEM)
N010 G92 X.5 Y.2 (SET PROGRAM ZERO)
N015 G90 (ABSOLUTE MODE)
N020 G41 H061 (LEFT OFFSET OF .0061)
```

N025 G51 A1.0 (TAPER LEFT WITH 1 DEGREE ANGLE)

N030 C411 (SELECT CUTTING CONDITION)

N031 T84 (HIGH PRESSURE FLUSH)

N032 T91 (AUTO THREAD)

N035 G01 X.485 (MOVE TO POINT 2)

N040 Y.015 (MOVE TO POINT 3)

N045 G03 X.5 Y0 I.015 (ARC IN APPROACH TO POINT 4)

N050 G01 X1. (MOVE TO POINT 5)

N055 Y1. (MOVE TO POINT 6)

N060 X0 (MOVE TO POINT 7)

N065 Y0 (MOVE TO POINT 8)

N070 X.4 (MOVE TO "GLUE STOP" POINT)

N075 M00 (STOP TO SUPPORT PART)

N080 X.5 (MOVE BACK TO POINT 4)

N085 G03 X.515 Y.015 J.015 (ARC OUT TO POINT 9)

N090 G01 Y.2 (MOVE TO POINT 10)

N095 G40 G50 X.5 (CANCEL RADIUS COMP AND TAPER, MOVE)

N100 M02 (END OF PROGRAM)

This program needs a little explanation. Blocks N035 to N045 and N085 to N095 allow the wire to instate and cancel compensation and offset on a "dummy move". This does two things. First, you know that wire radius compensation, when it is instated, may make an angular move to initiate. For punches this is fine, since the approach is in the glue stop area and is going to be ground off on a surface grinder anyway. But for a die, there will be no subsequent grinding operation. The entire inside shape must be smooth and cannot have any witness marks where the wire approaches the interior wall. So, instating offset and taper in the dummy move insures that the proper taper and offset will already be initiated when the wire makes its first move to the actual surface of the part to cut.

Second, the arc in and arc out movements to and from the die surface also insure that there will be minimal witness marks in this area. The wire will form a smooth approach to the first surface to cut.

Some operators will approach straight to the first surface to cut, go all the way around the die, and over travel the approach point by .050 inch or so. While this does approximately the same thing as arc in and arc out there may be witness marks in the approach area, since the wire will be attempting to recut the .050 over travel area.

Blocks N070 and N075 are necessary (especially for heavier parts) to allow the operator a chance to apply clamps, magnets, or glued tabs on the slug that is going to fall when the part is cut off.

These techniques are also explained during the program formatting section of the course.

TAPER SETTINGS IN THE CONTROL

As you can see, there really isn't all that much to programming taper in a die. You simply instate taper, use it, and cancel it. However, there are some machine settings that must also be made. You can find these settings in the SET screen in the lower right corner. If these settings are not properly made, the resulting taper will be unpredictable. The control needs to know five basic things about your setup in order to form the taper correctly. They include:

- 1) Table to limit (T-Limit)
- 2) Table to upper (T-Upper)
- 3) Table to lower (T-Lower)
- 4) Table to program (T-Program)
- 5) Table To next (T-Next)

"Table" here refers to the top surface of the pedestals. Figure 8-3 shows these relationships. Detailed description are on the following pages.

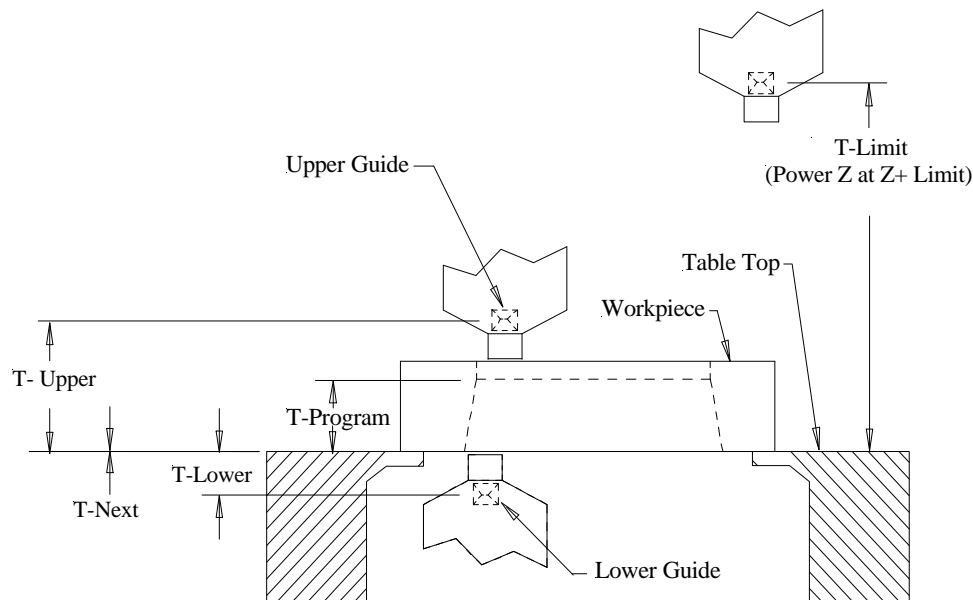


Figure 8-3

TABLE TO LIMIT

Table To Limit is handled differently for a manual Z-axis and a power Z-axis. In both cases, Table To Limit is a CONSTANT that will usually not change for the life of the machine. Like Table to lower, you will find it documented in the mechanical file printout

that comes with your machine, if not already input into the “set” screen. It is wise to write down the value of Table To Limit and keep it (with Table To Lower) posted on the machine. This value should only change if replacing the upper guide, or if the upper head gets crashed.

TABLE TO UPPER

Table to upper is the distance between the worktable and the upper guide. To correctly input this value, you have to know the value of Table To Limit for your machine (located on the “set” screen). Use the following information for setting T-Upper.

Table To Upper and Table To Limit For A Manual Z Axis Machine

Table To Limit for a manual Z axis machine is the distance from the tabletop to the upper guide WHEN THE VERNIER SCALE FOR THE Z AXIS HEIGHT IS READING ZERO.

Table to Upper value is known by positioning the Z-axis to the where you are going to position the head during cutting. Now, add the value on the vernier scale to the Table To Limit value, then input this sum into the Table To Upper register in the SET screen.

Table To Upper and Table To Limit For Power Z Axis Machines

Table To Limit for a power Z axis machine is the distance from the tabletop to the upper guide WHEN THE Z-AXIS IS AT ITS UPPER LIMIT SWITCH POSITION.

Table To Upper is automatic as the Z axis is moved up or down. The value of Table To Upper changes by itself because the machine knows where the head is at. Just be sure that this feature is turned on. Go to the SET, TRAVEL screen and look at the right column and verify that “*AUTO T UPPER*” is turned on.

Note: For both power or manual z axis machines Table To Limit is a machine constant and is not altered from job to job. It may need to be changed but only if your machine is not cutting taper accurately. To calibrate Table To Limit see the end of this chapter describing calibrating T-Upper T-Lower.

TABLE TO LOWER GUIDE

This is the distance from the worktable to the lower wire guide. This is a constant that will usually not change. Sodick documents this value in the list of parameters (the mechanical file) that comes with your machine. You should write this value down and keep it near the machine. NOTE: This is a positive number.

TABLE TO PROGRAM

This is the distance from the tabletop to where you are holding size in X Y.

When taper cutting (G51 / G52)

Here are four possible situations when the workpiece is mounted directly on the table.

- 1) Cutting with the top of the die UP and you want a die land, this value will be the thickness of the part MINUS the die land.
- 2) If cutting with the top of the die UP and you do NOT want a die land, this value will be the thickness of the part.
- 3) If cutting with the top of the die DOWN and you want a die land, this value will be the die land amount.
- 4) If cutting with the top of the die DOWN and you do NOT want a die land, this value will be zero.

When 4-Axis cutting (G74)

When 4 axis cutting (G74 or G141) this is the amount that the bottom of the part is above the worktable. In this case a fixture would be lifting the part from the worktable.

TABLE TO NEXT

This parameter sets the opposite side of the part from T-Program, in respect to the worktable.

When taper cutting (G51 / G52)

When taper cutting, Table to Next will be zero if the die land is up. If the die land is down (T-Program = 0) then T-Next is the workpiece thickness.

When 4-Axis cutting (G74)

This value represents the distance from the table to the 2nd plane when you have a 4 axis (independent UV) program. When the part is on the table, Table to Program is usually 0 (1st plane) and Table to Next will usually be the workpiece thickness (2nd plane).

Note: When 4 axis cutting, it must be used. This sets the position of the u-v axis plane in your part, so set this value to the amount that the top of the part is above the table.

TABLE TO DRAW

This value is not used during cutting or dry run 1 or 2 or in graphics. But in DRY RUN 3, it will tell the control what surface you wish to see on the plotting table. For example, if the value in Table to Draw is zero, you will see what the bottom of the part will look like. If the value in Table to Draw is 2.00, you will see what the surface would look like two inches above the tabletop. The best application for this is when you have a program that has two different shapes on the top and bottom of the part (independent UV programming), or if you have a taper program and want to be sure everything is set correct.

NOTES ABOUT DRY RUN

Always dry run the program when all axis are in the proper startpoint location. This is your last chance to catch any mistakes. Here is how Dry Run works in its three forms:

DRY RUN MODES

Dry Run 1: No sensor touch. If the wire hits something, it will break. Used for most Dry Run tries. All axes motors run, all limit switches are checked.

Dry Run 2: Sensor touch enabled. If the wire touches something, an alarm will sound. Used if you are trying to insure that the wire will not cut into a clamp, fixture etc. All axis motors run, all limit switches are checked.

Dry Run 3: Table to Draw enabled (see page 8-9). The “z plane” being drawn by the motion of the x-y axis on the plotting table will be specified by Table to Draw. Only X and Y-axes motors move in Dry Run number 3.

PARTIAL DRY RUN

If you want to Dry Run through a portion of the program and then switch to cutting, turn “Dryrun” to 1 and press enter, then turn “Dryrun” to 0 as the machine is traveling on the line before the line you want to start cutting on.

DRY RUN SPEED

The Dry Run speed can be controlled by the selection of the “MFR” 1 to 3 key. These keys work the same for Dry Run as they do for jogging the machine, 0 = fastest, 3 = slowest.

DRY RUN AND LIMIT SWITCHES

While Dry Running your job, if an axis travels beyond its limit, the machine may not stop with the “Limit Switch” error. This is a normal condition do to the speed of the Dry Run. It is required by the operator to monitor the axis position on the “SET” “DISPLAY”

screen, to be sure this doesn't occur. Otherwise the machine may stop while machining with the error. "Limit check" in graphics can also be used to check for overtravel.

TAPER SETTING EXAMPLES

Below are two examples showing "SET" screen taper setting values with a manual Z-axis.

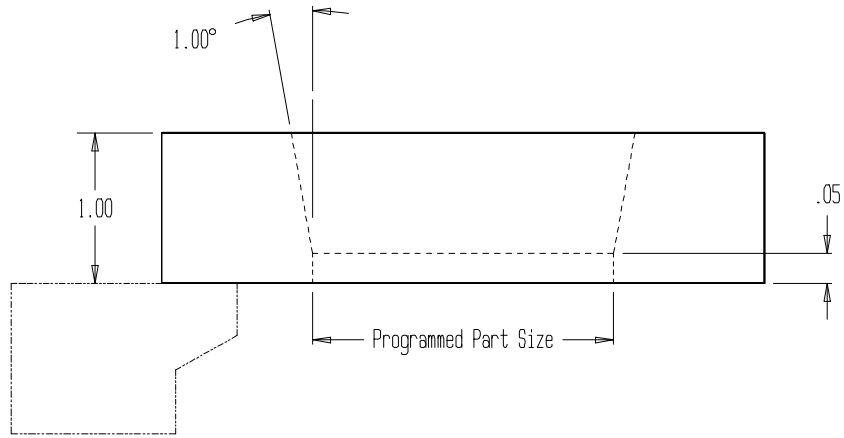


Figure 8-4

<i>Known Values</i>	<i>Operator input</i>
T-Limit = .6990"	T-Upper = 1.5740"
T-Lower = .3535"	T-Program = .0500"
Z Vernier = .8750"	T-Next = 1.000

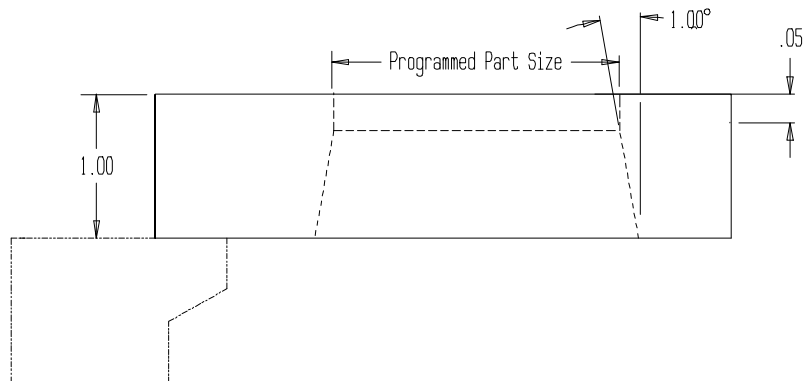


Figure 8-5

<i>Known Values</i>	<i>Operator input</i>
T-Limit = .6990"	T-Upper = 1.6740"
T-Lower = .3535"	T-Program = .9500"
Z Vernier = .9750"	T-Next = 0.000

TAPER SETTINGS IN BRIEF – OPERATOR INPUT

The following is what needs to be actually input by the operator for each taper cutting job. For complete parameter definitions see prior info.

MANUAL Z AXIS MACHINES

- 0 Input Table To Upper (Z vernier + T-Limit)
 - 2) Input Table To Program (Distance to where you are holding size from the worktable)
 - 3) Table To Next (set to zero if T-Program is > 0)
- All other settings are machine constants and should already be set.

POWER Z AXIS MACHINES

- 0 Input Table To Program (Distance to where you are holding size from the worktable)
 - 2) Input Table To Next (set to zero if T-Program is > 0)
- All other settings are machine constants and should already be set.

TAPER SETTINGS IN BRIEF FOR 4 AXIS CUTTING (G74-G141)

MANUAL Z AXIS MACHINES

- 0 Input Table To Upper (Z vernier + T-Limit)
 - 2) Input Table To Program (Distance from worktable to bottom of part – X Y Axis Plane)
 - 3) Input Table To Next (Distance from worktable to top of part)
- All other settings are machine constants and should already be set.

POWER Z AXIS MACHINES

- 0 Input Table To Program (Distance from **worktable to bottom** of part – X Y Axis Plane)
 - 2) Table To Next (Distance from **worktable to top** of part)
- All other settings are machine constants and should already be set.

Note: Now Table To Next is actually being used in 4-axis cutting. Its value should represent the workpiece thickness + fixturing thickness if one is being used.

NOTES CONCERNING TAPER CUTTING

CALIBRATING T-UPPER AND T-LOWER

If you are not cutting accurate taper angles, it is possible that the constants called Table to Limit and Table to Lower in the SET screen need to be changed. There are programs called Taprdata or Tdstrok3 or Tdstrok located on your User Disk or Harddrive. These programs will calculate the 2 constants. Locate one of these program names and load the program into memory then follow the instructions at the beginning of the program. It is wise to do a test cut after this procedure is done, to verify that the machine is now cutting accurate taper angles. This is a procedure that is not done very often. If a different wire guide is installed, you may want to run one of the programs mentioned above to recalculate the constants.

NOTE: TAPRDATA is the latest program to date as of Feb. 1998.

FLUSH NOZZLES

The standard flush cups have a 6mm I.D. and are used when cutting up to 10° taper. If 11° to 20° is being cut, flush cups with a larger I.D. (10mm) should be used. If 21° to 30° is being cut, flush cups with a larger I.D. (12mm) should be used. Verify by dry running the machine, that the wire does not contact the edge of the hole on either flush cup. A mirror may be needed to check the upper nozzle.

0 to 10°	6mm Lower (3081674)	6mm AWT Upper (3081604)
11° to 20°	10mm Lower (3081675)	10mm AWT Upper (3081606)
21° to 30°	12mm Lower (3081867)	12mm AWT Upper (3081868)

WIRE TYPE CONSIDERATIONS

At steep angles, if there are lines on the part from wire vibration, try using a soft wire like Gisco Mega cut W, or Gisco Berco cut 390 to eliminate the lines. These wires have more elongation and will flow through the wire guides better at taper angles of more than 10 degrees.

CONVERTING TO DECIMAL DEGREES

To convert the minutes and seconds of an angle given in degrees, minutes, seconds to the decimal equivalent (without this function on your calculator).

Decimal equivalent = [Min / 60] + [Seconds / 3600]

GRAPHICS COLORS

When drawing or viewing the graphic picture of a program with taper, the screen will show 2 colors. While running a taper program, there will also be 2 asterisks following the graphics drawing.

Blue = Top of workpiece

Green = Bottom of workpiece

USE THESE PAGES FOR NOTES

CHAPTER 9 PROGRAM FORMATTING

The fifth Key Concept to operation is that you must understand the formatting of a CNC wire EDM program. We have already looked at several complete programs, but they were limited in their application. In this section of the course, we will show you more good techniques for EDM programming.

The techniques and programming codes discussed so far may not make total sense, but as you write more programs and practice what you have learned, this will get better. Also, remember you have the conversational APT programming software that is built into the control to minimize the need to use manual programming techniques.

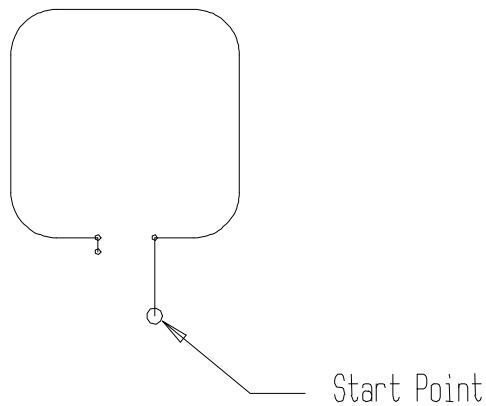
Now, let's talk about program formatting. As you have seen in the examples shown so far, there aren't that many different words and codes used in a program and you may already be familiar with some of them.

Before we give examples of program formatting, we want to discuss the methods by which different types of workpieces are machined on a wire EDM machine. You may also want to review the information given so far.

As you know, accuracy and finish are directly related to the number of trim passes being made. In some cases one pass around the shape will be adequate when accuracy and finish are not critical. But when trim passes must be made you must understand how to do them correctly for the various types of parts you will be machining. We will discuss several types of workpieces that are typically machined on a wire EDM machine and try to present the most important considerations.

PUNCHES

Punch machining requires that an area of the shape be left "UN-machined". This area is called the tab area. The purpose for the tab area is to allow the punch to remain attached to the main workpiece during trim passes. The size of the tab is directly related to the size of the part and the tab should be located so it remains stable while it is supporting the part. Also the location of the tab area should be planned in a way to allow easy grinding of the tab that will be left after the punch is cut off. The figure 9-1 is a drawing that shows the programmed movements for a typical punch BEFORE THE CUT OFF.

*Figure 9-1*

After the punch is machined the operator will tell the control to stop (with an M00). This will allow the operator to apply clamps, magnets etc. These will support the punch during the cutoff. Then the operator will command the punch to be cut off with a straight cut across the tab area. It is wise to cut the punch off in a way that leaves a minimum amount of stock for grinding (.010" stock or so).

If trim passes are required on a punch, it makes programming more difficult. This is because it is usually desired that the trim passes alternate forward and reverse. This allows all the passes to be performed without stopping the machine to rethread the wire. In this method, the NC program would need to be written to include the reverse motion.

DIES

Dies present a slightly different problem. On the first rough pass, the machine must be stopped just prior to when the slug is to be cut off; otherwise the slug will interfere with the movement back to the start point. The operator then supports the part and the cut-off is executed. It must be stopped again after the cut off before the wire returns back to the start point. This will allow the operator to remove the slug.

If trim passes are necessary, it is best to make them in a reverse direction (but not mandatory). If the trim passes are made in the same direction as the rough pass the wire will have the tendency to follow the motions of the rough pass. If any washout occurred on corners during the rough pass, the trim pass will try to follow the washout. But if reverse direction trim passes are made, the wire will have the tendency to start fresh as it enters any washout areas. However, reverse trim passes are a little more difficult to program (manually) and many manual programmers will simply program trim passes in the same direction as the rough pass.

FORM TOOLS

With a form tool, the cutting path is not in an enclosed shape. Generally, a form tool is cut from one end to the other. During the rough pass, the wire could be stopped just prior to reaching the other end of the shape. This will allow the operator to support the slug end of the stock as it is cut loose. If trim passes are necessary, after the rough pass is completed another stop is made to allow the operator to remove the slug.

If trim passes are necessary, it is wise to make them in a reverse direction for the same reasons as for a die. But if you want to make the trim passes in the same direction as the rough pass (because it is easier), you can rapid (G00 code) the wire back to the start point and make the first trim pass. This can be repeated for as many trim passes as necessary.

WORDS INVOLVED IN SUBROUTINES

Before we give examples of each type of shape, we want to show you how to develop subroutines in a way that simplifies making trim passes. Subroutines keep you from having to write redundant commands a second time, and basically trim passes are redundant commands. Your goal with subroutines will be to isolate the trim passes from the main program in a way that lets you simply command that the trim pass be made when you want them. At first this might seem a little complicated, but when you see an example you will realize that it really is straightforward.

SUBROUTINES INVOLVE FOUR PROGRAMMING WORDS:

M98 - **Jump** to a sub routine

M99 - **Return** to the main program

P word - **Sequence** line number to jump to?

L word - **Loop** the sub routine how many times?

The M98 command will ALWAYS include a P word to tell the control where to go. When the control reads **M98 P1000** for example, it jumps to sequence number **N1000** in the subroutine and continues executing from there. When the control reads an M99 command, it jumps back to the main program to the command AFTER the M98 P1000 and continues execution. The commands that are between the N word and the M99 will include the movement commands (G codes) to drive the wire around the shape.

The subroutine (beginning with the N word, which is specified by the P word) will ALWAYS be included AFTER the end of the main program (after the M02 of the main program).

At this point, let's look at some examples.

PUNCH EXAMPLE

Say you were machining a punch that required three total passes (rough and two trims). Based on the Material Type, Material Thickness, Wire Diameter and Flushing condition you looked in the Sodick condition manual and below are the set of conditions and offsets the manual recommended:

Rough Pass - C411 and H082

First Trim Pass - C621 and H056

Second Trim Pass - C651 and H053

Here is the print for the punch:

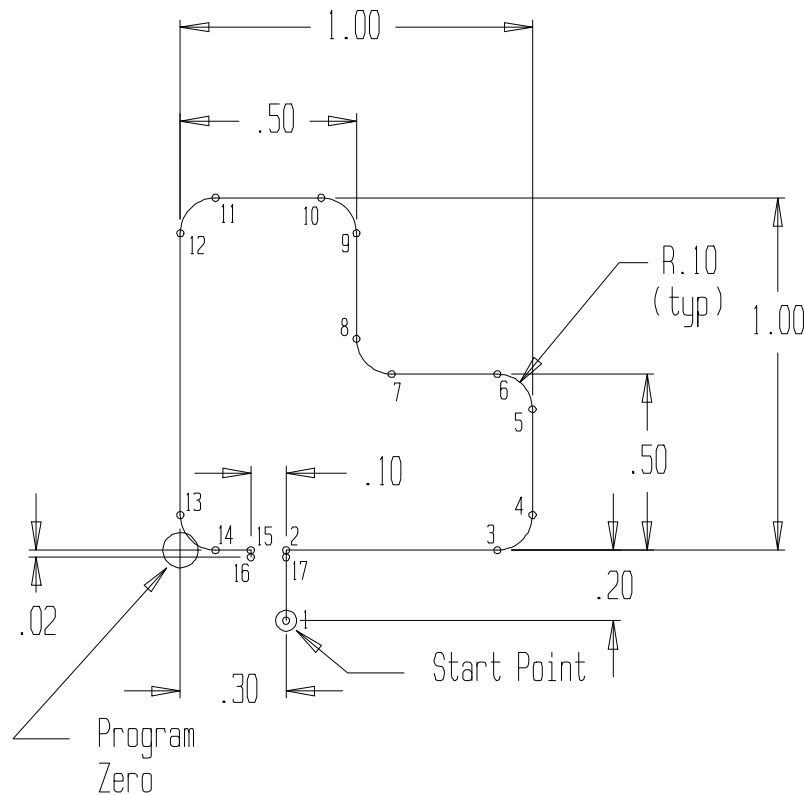


Figure 9-2

Chapter 1 NC Program for figure 9-2 Using Sub Routines

N005 G54 (SELECT COORDINATE SYSTEM IN MAIN PROGRAM)

N010 G92 X.3 Y-.2 (SET PROGRAM ZERO)
N015 G90 (SELECT ABSOLUTE MODE)
N020 C411 (SELECT CONDITION FOR ROUGH PASS)
N021 T91 (AUTO WIRE THREAD)
N025 G42 H082 (.0082" OFFSET TO THE RIGHT)
N026 T88 (SUBMERSE FOR SUBMERSIBLE MACHINES ONLY)
N027 T84 (TURN ON HIGH PRESSURE FLUSHING PUMP)
N030 **M98 P1000** (JUMP TO N1000 AND MAKE ROUGHING PASS)
N033 T85 (TURN OFF HIGH PRESSURE PUMP)
N035 C621 (SELECT CONDITION FOR FIRST TRIM PASS)
N040 G41 H056 (SEL. COMP LEFT & .0056 OFFSET, 1ST TRIM PASS)
N045 **M98 P2000** (JUMP TO N2000 AND MAKE FIRST TRIM PASS)
N050 C651 (SELECT CONDITION FOR SECOND TRIM PASS)
N055 G42 H053 (SET .0053 OFFSET TO RIGHT SECOND CUT)
N058 **M98 P1000** (JUMP TO N1000 AND FOR 2ND TRIM CUT)
N060 M00 (STOP TO APPLY MAGNETS FOR CUTOFF)
N065 C411 (SELECT ROUGHING CONDITION FOR CUTOFF)
N066 T84 (TURN ON HIGH PRESSURE FLUSHING PUMP)
N070 G01 X.3 (CUT OFF TO POINT 17)
N075 Y-.2 (MOVE BACK TO START POINT)
N080 M02 (END OF MAIN PROGRAM)

(COUNTER CLOCKWISE SUB ROUTINE)

N1000 G01 Y0 (MOVE TO POINT 2)
N1005 X.9 (MOVE TO POINT 3)
N1010 G03 X1. Y.1 J.1 (MOVE TO POINT 4)
N1015 G01 Y.4 (MOVE TO POINT 5)
N1020 G03 X.9 Y.5 I-.1 (MOVE TO POINT 6)
N1025 G01 X.6 (MOVE TO POINT 7)
N1030 G02 X.5 Y.6 J.1 (MOVE TO POINT 8)
N1035 G01 Y.9 (MOVE TO POINT 9)
N1040 G03 X.4 Y1. I-.1 (MOVE TO POINT 10)
N1045 G01 X.1 (MOVE TO POINT 11)
N1050 G03 X0 Y.9 J-.1 (MOVE TO POINT 12)
N1055 G01 Y.1 (MOVE TO POINT 13)
N1060 G03 X.1 Y0 I.1 (MOVE TO POINT 14)
N1065 G01 X.2 (MOVE TO POINT 15)
N1070 G40 Y-.02 (CANCEL OFFSET AND MOVE TO POINT 16)
N1075 M99 (END OF SUB ROUTINE, JUMP BACK TO MAIN)

(CLOCKWISE SUB ROUTINE)

N2000 G01 Y0 (MOVE TO POINT 15)
N2005 X.1 (MOVE TO POINT 14)
N2010 G02 X0 Y.1 J.1 (MOVE TO POINT 13)
N2015 G01 Y.9 (MOVE TO POINT 12)
N2020 G02 X.1 Y1. I.1 (MOVE TO POINT 11)

N2025 G01 X.4 (MOVE TO POINT 10)
N2030 G02 X.5 Y.9 J-.1 (MOVE TO POINT 9)
N2035 G01 Y.6 (MOVE TO POINT 8)
N2040 G03 X.6 Y.5 I.1 (MOVE TO POINT 7)
N2045 G01 X.9 (MOVE TO POINT 6)
N2050 G02 X1. Y.4 J-.1 (MOVE TO POINT 5)
N2055 G01 Y.1 (MOVE TO POINT 4)
N2060 G02 X.9 Y0 I-.1 (MOVE TO POINT 3)
N2065 G01 X.3 (MOVE TO POINT 2)
N2070 G40 Y-.02 (CANCEL OFFSET AND MOVE TO POINT 17)
N2075 M99 (END OF SUB ROUTINE, JUMP BACK TO MAIN)

The subroutine techniques shown in the previous example are almost identical to the techniques that the conversational APT system uses. So, even if you are going to be inputting conversational programs, and seldom writing manual programs, this will give you an insight into how the conversational side will generate CNC programs for you.

Notes about the punch program:

1) The conditions and offsets were for 1/2" thick steel using 010" wire. You MUST check in your condition manual to obtain the correct conditions and offsets for your application.

2) Note that the main program (through the M02) will be very similar from one program to the next. Only the subroutines will change dramatically from one punch program to the next.

3) Parentheses are allowed in the program at any time to allow you to include a message to the operator.

Example: N0055 MOO (SUPPORT THE PART AND PRESS RST)

4) A CRT command is available to echo a note onto the CRT while the machine is running.

Example: N0009 CRT (ROUGH CUT JOB #1)

NOTE: maximum of 24 characters inside()

DIE EXAMPLE

Figure 9-3 shows a die opening. In this example, we make two total passes, one rough for the tapered area and one skim for the land (tapered area first). Note that this is the die opening for the previous punch example. While two passes are being made, we make them in the same direction. We are also pulling a fast one and tricking the control with the optional block skip technique for the tab area during the roughing pass. See if you can figure out what is happening.

NOTE: We are using a 1 cut on size offset for the taper cut, since no trims will be done on the tapered area.

Say you looked in the condition manual and found the conditions and offsets to be as follows:

Rough pass (with taper) - C411 and H062

Trim pass (straight portion) - C620 and H053

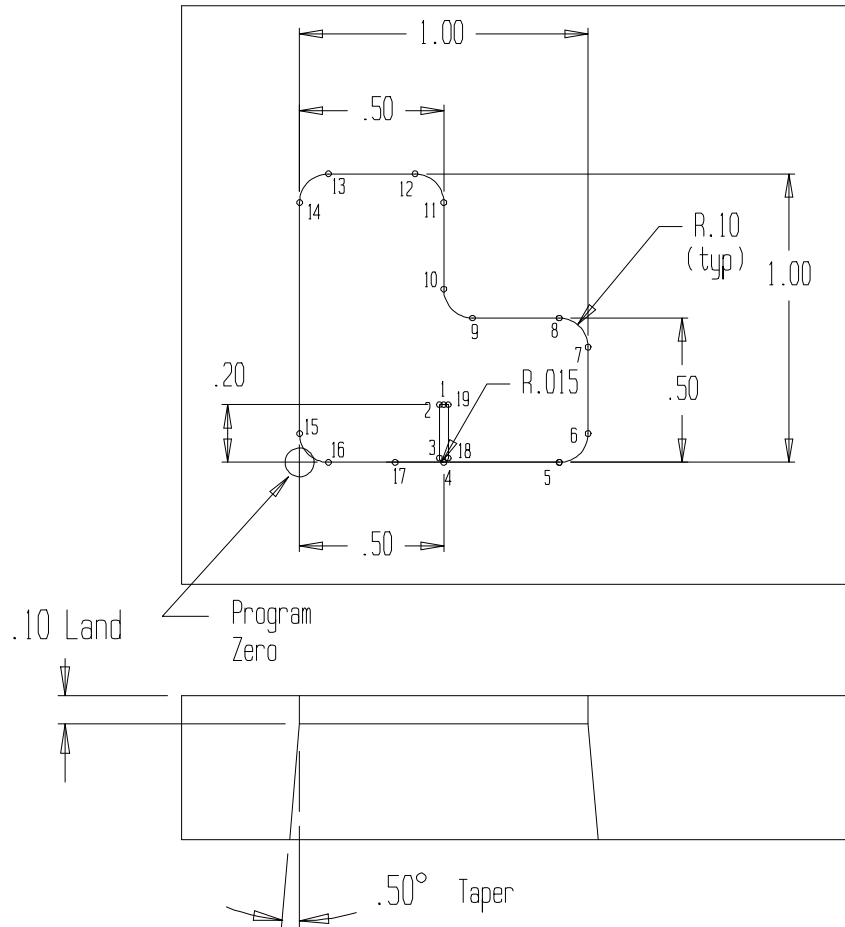


Figure 9-3

NC Program for figure 9-3

N005 G54 (SELECT COORDINATE SYSTEM)

N010 G92 X.5 Y.2 (SET PROGRAM ZERO)

N015 G90 (SELECT ABSOLUTE MODE)

N020 C411 (SELECT CONDITION FOR ROUGH PASS)

N021 T91 (AUTO WIRE THREAD)
N025 G41 H062 (COMP LEFT AND .0062 OFFSET 1ST CUT)
N026 G51 A.5 (SET TAPER TO LEFT, .5 DEGREE)
N027 G12 (TURN OPTIONAL BLOCK SKIP OFF TO ACTIVATE GLUE STOP)
N028 T94 (SELECT SUBMERSE MODE FOR SUBMERSIBLE
MACHINES ONLY)
N029 T84 (TURN ON HIGH FLUSH PUMP)
N030 **M98 P1000** (JUMP TO N1000 AND MAKE ROUGHING PASS)
N033 T85 (TURN OFF HIGH PRESSURE PUMP)
N035 C620 (SELECT CONDITION FOR STRAIGHT PORTION TRIM
PASS)
N040 G41 H053 (SELECT COMP LEFT AND .0053 OFFSET FOR
STRAIGHT PORTION TRIM PASS)
N043 G11 (TURN OPTIONAL BLOCK SKIP ON TO IGNORE GLUE
STOP)
N045 **M98 P1000** (JUMP TO N1000 AND MAKE STRAIGHT
PORTION TRIM PASS)
N050 M02 (END OF MAIN PROGRAM)

(COUNTER CLOCKWISE SUB ROUTINE)

N1000 G01 X.485 (MOVE TO POINT 2)
N1005 Y.015 (MOVE TO POINT 3)
N1010 G03 X.5 Y0 I.015 (ARC IN TO POINT 4)
N1025 G01X.9 (MOVE TO POINT 5)
N1030 G03 X1. Y.1 J.1 (MOVE TO POINT 6)
N1035 G01 Y.4 (MOVE TO POINT 7)
N1040 G03 X.9 Y.5 I-.1 (MOVE TO POINT 8)
N1045 G01 X.6 (MOVE TO POINT 9)
N1050 G02 X.5 Y.6 J.1 (MOVE TO POINT 10)
N1055 G01 Y.9 (MOVE TO POINT 11)
N1060 G03 X.4 Y1. I-.1 (MOVE TO POINT 12)
N1065 G01 X.1 (MOVE TO POINT 13)
N1070 G03 X0 Y.9 J-.1 (MOVE TO POINT 14)
N1075 G01 Y.1 (MOVE TO POINT 15)
N1080 G03 X.1 Y0 I.1 (MOVE TO POINT 16)
/ N1085 G01 X.4 (MOVE TO POINT 17)
/ N1090 M00 (STOP TO APPLY MAGNETS, NOTE “/” CODE)
N1095 G01 X.5 (MOVE TO POINT 4)
/N1100 M00 (Stop To Remove Slug)
N1105 G03 X.515 Y.015 J.015 (ARC OUT TO POINT 18)
N1110 G01 Y.2 (MOVE TO POINT 19)
N1115 G40 G50 X.5 (CANCEL COMP ,TAPER & MOVE TO POINT 1)
N1120 M99 (END OF SUB ROUTINE)

Notes about the die program:

- 1) Always check the condition manual to obtain the correct conditions and offsets for your application and machine tool type.
- 2) The same comments concerning parentheses, which were made for the previous (punch) program also, apply here.
- 3) The techniques shown in this program are very similar to the way that the conversational APT system will do it. The only changes are that the APT system will NOT use the optional block skip technique. It will create THREE subroutines. Also, the APT system will create reverse direction trim passes for you (the better way to do it).

CHAPTER 10 DESCRIPTION OF OPERATING AIDS

The sixth and last Key Concept is that you have operating aids available to simplify programming and operating the machine. There are more capabilities you need to know about to make running the machine easier. Here is a list of the programming aids that we will look at in this section of the course:

- 1) “Q” Routine Techniques
- 2) G and M codes for Q routines
- 2) Coordinate System Shifting (G54-G959)
- 3) Mirror Image and Axis Exchange
- 5) Sub-Routines
- 6) Scaling
- 4) Automatic Corner Rounding
- 7) Axis Rotation
- 8) String File Techniques

When you know the powers of these features, it should drastically reduce your long term programming effort. Pay close attention to the discussion of “Q” routines. Almost every company utilizing our wire EDM equipment can benefit from their use.

Q ROUTINE TECHNIQUES (MACROS)

We begin our discussion of programming aids with Q routines or macros since they can be combined with other programming aids to make programming easier. You should understand how Q routines work. Q routines are a powerful feature of the control.

WHAT IS A Q ROUTINE?

A Q routine is a program that automates something or simply cuts a shape. A Q routine is a program like the centerfind and corner find programs.

A Q routine is similar to any other program you have seen so far with one exception. A Q routine has no special ending word. As you know, main programs end with M02 and sub-routines end with M99. A Q routine ends with no special program ending word. They end with the last command being done in the Q routine.

The best way to get the idea is to compare Q routines to sub-routines (introduced during concept number five.) As you know, a sub- routine is included in the same file as the main program but is placed after the M02 of the main program. A sub-routine is totally

redundant; meaning the control will simply execute the sub- routine command by command, in the same fashion each time the sub-routine is called up.

However, a Q Routine does not have to be part of another program. That is, the Q routine will be a separate file or program. A Q routine can be called up at any time from another program, or from the MDI mode. Also, Q routines allow variables to be passed from the MDI mode or from a call statement included in another program. You can also do arithmetic calculations in a Q routine.

MACHINE START-UP Q ROUTINE

Whenever the machine is started up, you should do several things to complete the start-up procedure. After turning on the power, you have been:

- 1) Sending the machine to its limits in X, Y, U, V and Z.
- 2) Checking the display screen to be sure machine position shows 0 on the XYUVZ axis.
- 3) Sending the U and V axis back to vertical.

These can be tedious commands to execute every time you source on the machine. Let's look at a simple Q routine that could be made, to make homing the machine easier.

(Q ROUTINE NAMED "STARTUP")

N005 G81 X- Y- U- V- Z- (SENDS AXES TO MINUS LIMITS)

N010 G00 UV (SENDS U AND V AXIS BACK TO ZERO)

That's the entire Q routine program. Notice again that this program ended with no special command. Once you type it in, you need to save it to the user disk or harddrive. When you save this program, it can be any name you wish. For example, you named this program STARTUP.

In the MAL, MDI mode you type this command to execute the Q routine.

QSTARTUP

When the control executes the *Q STARTUP* command, it will load the program named STARTUP into memory and run it. In this program the machine will travel to its limits and move U and V back to zero. All of this will be done automatically. So, if you have this program named STARTUP on your user disk or harddrive, it makes machine homing easier.

VARIABLES IN Q ROUTINES

So far, we have just scratched the surface of what can be done with Q routines. You also

have the ability to pass variables to the Q routine and have the Q routine behave differently based on the values of the variables.

Parentheses after Q filename can be added to allow the passing of values into the header of the Q'ed file. If you made the command QOVAL(.5000,1.0000) the values .5000 and 1.0000 would be passed to registers H000 and H001 in the header of the NC program named OVAL. Notice the comma between the variables. As many variables as you need can be passed from the call statement (all separated by commas) to the actual NC program.

To understand where the passed variables will go, let's look at the format of the header of a program. All programs will begin with a header (created by NEWFILE) that includes a series of H registers that look like this:

```
H000 = +000000000  H001 = +000000000  H002 = +000000000  
H003 = +000000000  H004 = +000000000  H005 = +000000000
```

We previously used these H registers when we needed to input offset values. If more than 6 registers are needed, you can simply type them in. The FIRST variable being passed (the extreme left variable) will be placed in the first H register. In our case, it would be placed in H000. The second value from the call statement is placed in H001, the third in H002, and so on.

In our previous example, we gave the command QOVAL(.5000,1.0000). In this case the value .5000 would be placed in H000 and the value 1.0000 would be placed in H001.

In the Q routine, any time you want to reference the values being passed, you can refer to the H register in your NC program. G01 X+H001, this would cause the machine to cut in the X axis to the value stored in the H001 register.

An example of this would be, you want to pass the condition number and offset value to the Q routine. Here's how the call statement could look:

```
QEXAMPLE(411,061)
```

Where the 411 is the cutting condition number to be passed and the 061 is the offset value to be passed to the Q routine named EXAMPLE.

(Q routine named example)

```
H000 = +000000000 H001 = +000000000 H002 = +000000000  
H003 = +000000000 H004 = +000000000 H005 = +000000000 (Continued on next page)
```

N005 G54
N010 G92 X.5 Y-.25
N015 G90
N020 C[H000]
N025 G42 H[H001]

In block number N020, the C word is specified by a variable. Whatever condition number was passed to the Q routine would be placed in H000 and this value is being referenced in the Q routine in block number N020. Notice the brackets around the H000 in block N020. These are NOT parentheses. They are square corner brackets. The keyboard on your control has these characters. The brackets MUST isolate the H words being referenced in the Q routine. In this manner the control will add the value which was transferred into H000 to the C code called.

In block number N025, the offset being set by the second variable (H001) will be passed from the call statement into the program. Please note that brackets are not really necessary because the H word by default is the offset word, so the block could really be G42 H001.

ARITHMETIC IN A Q ROUTINE

Another feature of Q routine techniques is that you have the ability to do arithmetic calculations right in the Q routine. This means that by using arithmetic with the variables being passed, you can create a general purpose Q routine that will behave differently each time it is called (when different variables are passed.)

Arithmetic calculations are used primarily to calculate axis coordinates and radius values. ALL arithmetic calculations MUST be enclosed in square brackets [].

Here is an example of a command that includes a calculation with variables:

G01 X[H000 / 2]

This command would cause the wire to cut to a position in X of half of whatever value is stored in variable number H000. You can also combine arithmetic calculations in one command. For example, say the current value of H000 is 5000 (.5 inch) and H001 is currently set to 10000 (1 inch). See if you can evaluate the result of this command:

X-[[H001/2]-[H000/2]]

The priority of any combined arithmetic expression is like that of any calculator. That is, anything in brackets will be done first, then multiplication and division, and last, addition and subtraction.

In the above command, first the value of H001 (10000) would be divided by two (result =

5000). Then the value of H000 (5000) will be divided by two (result = 2500). And last, the control will subtract 2500 from 5000 and the result of this combined expression will be 2500 (.25 inch).

These simple commands by themselves don't mean much. But when they are used with a completed Q routine properly, there are fantastic implications. There is an example that shows you the real power of the Q routine techniques on the following pages.

OVAL SHAPED DIE BUTTON EXAMPLE

One excellent application for Q routines is with die buttons. Companies that machine die buttons usually have a limited number of shapes to be machined in the die button, but there may be many different sizes of the same basic shape. This is when variables and Q routines come to help out. A program can be written using these techniques which allows the input of sizes so the operator can adjust the NC program to cut the size part that is needed.

Here is a drawing that shows an oval shaped die button.

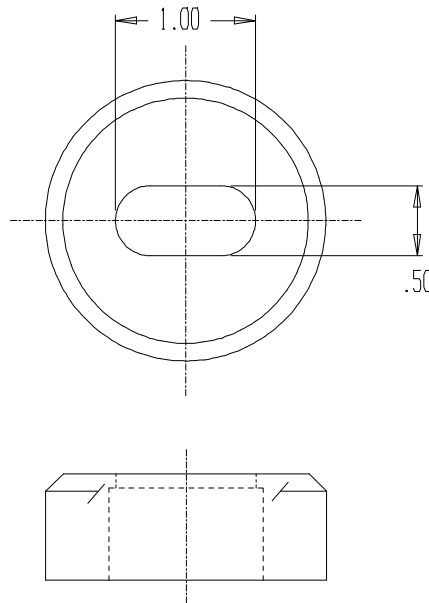


Figure 10-1

Wouldn't it be nice if all you had to do to machine this die button is make the command:

`QOVAL(5000,10000)`

Believe it or not, this is possible. However, let's also add the condition and offset to be used to the call statement so the revised version looks like this:

QOVAL(.500,1.000,411,.0061)

Now, four values will be passed to the Q routine named OVAL. In this case, you would be making a die button that was .500 inch high and 1.00 inch long, using condition number C411, and offset .0061 inch. But you can also see how easy it would be to make a different size oval shaped die button if the Q routine named OVAL were properly written and stored on the user diskette or hard drive!

Figure 10-2 is a drawing that shows the motions that will be made in the Q routine: Note that this Q routine ASSUMES THAT THE WIRE IS POSITIONED MANUALLY TO THE CENTER OF THE OVAL FIRST.

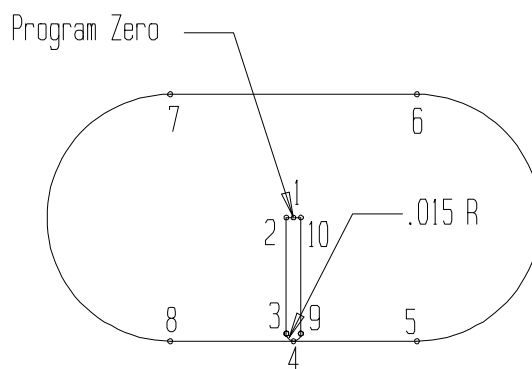


Figure 10-2

In the example below you could also enter the values into registers H 0,1,2 and 3.

NC Program for figure 10-2

H000 = +00000000 H001 = +00000000 H002 = +00000000
H003 = +00000000 H004 = +00000000 H005 = +00000000

N005 G54 (SELECT COORDINATE SYSTEM)
N010 G92 X0 Y0 (SET PROGRAM ZERO IN CENTER OF OVAL)
N015 G90 (SET ABSOLUTE MODE)
N020 C[H002] (SELECT CONDITION)
N025 G41 H[H003] (SET LEFT COMP, OFFSET VALUE)
N027 T88 (FOR SUBMERSIBLE MACHINES ONLY)
N028 T84 (TURN ON HIGH FLUSH PUMP)
N030 G01 X-.015 (MOVE TO POINT 2)
N035 Y-[H000/2]-.015] (MOVE TO POINT 3)
N040 G03 X0 Y-[H000/2] I.015 (ARC IN TO POINT 4)
N045 G01 X[[H001/2]-[H000/2]] (MOVE TO POINT 5)
N050 G03 Y[H000/2] J[H000/2] (MOVE TO POINT 6)
N055 G01 X-[[H001/2]-[H000/2]] (MOVE TO POINT 7)
N060 G03 Y-[H000/2] J-[H000/2] (MOVE TO POINT 8)
N065 G01 X0 (MOVE BACK TO POINT 4)

N070 G03 X.01 Y-[[H000/2/]-.015] J.015 (ARC OUT TO POINT 9)
N075 G01 Y0 (MOVE TO POINT 10)
N080 G40 X0 (CANCEL OFFSET AND MOVE BACK TO POINT 1)

Once you understand these techniques, it opens the door to many applications. To develop the Q routine, it may take longer than the amount of time it takes to write only one long hand program. But once the Q routine is written, programming each new die button will be very simple and quick.

G AND M CODES FOR Q ROUTINES

Along with the features shown to this point, Sodick has also designed some special G codes that have application with Q routine techniques.

G80 - Sensor Touch

This command can be used to pick up the edge of a workpiece. But to this point, you have only used the G80 in MAL, MDI mode. The G80 can also be used in a program. The center hole pickup Q routine (0145) and corner pickup Q routine (0165) use this G code. If you understand the techniques shown here, you will be able to develop special purpose pickup routines.

For example: G80X-

This tells the machine to move in the minus X direction until the wire touches the workpiece. Then the wire picks up the edge 3 times and the control takes an average and positions the machine accordingly.

M05 - Disable Wire Sensor

This command tells the control to ignore the fact that the wire is touching the part. Normally this condition would generate an alarm. But there are times in a program that you want the control to make a movement even though the wire is touching so program in an M05 along with the next movement. Example: M5X2.0

G82 - Split the Difference Command

This command tells the control to move the machine to half the value in the display. It is very helpful in the Q routine to pick up the center of a hole. G82X would move the machine to half the value in the X display on the CRT screen.

G83 - Transfer Current Position Into H Register

This command allows you to obtain the current position in X or Y and store it into a variable register. This can be helpful when developing special purpose pickup Q routines.

The command G83 X005 would load the current X position of the wire into the H005 register.

COORDINATE SYSTEM SHIFTING (G54-G959)

You know by now that Sodick allows up to sixty different coordinate systems to be used. These coordinate systems are helpful when you need to run multiple parts in a setup (like multiple die openings in a progressive die.). But there are also other applications for their use. Here we take a look at some applications for coordinate system shifting. We will be using some Q routine techniques, so please be sure you have read the Q routine descriptions in the beginning of this chapter.

EASILY PROGRAMMING MULTIPLE DIE OPENINGS

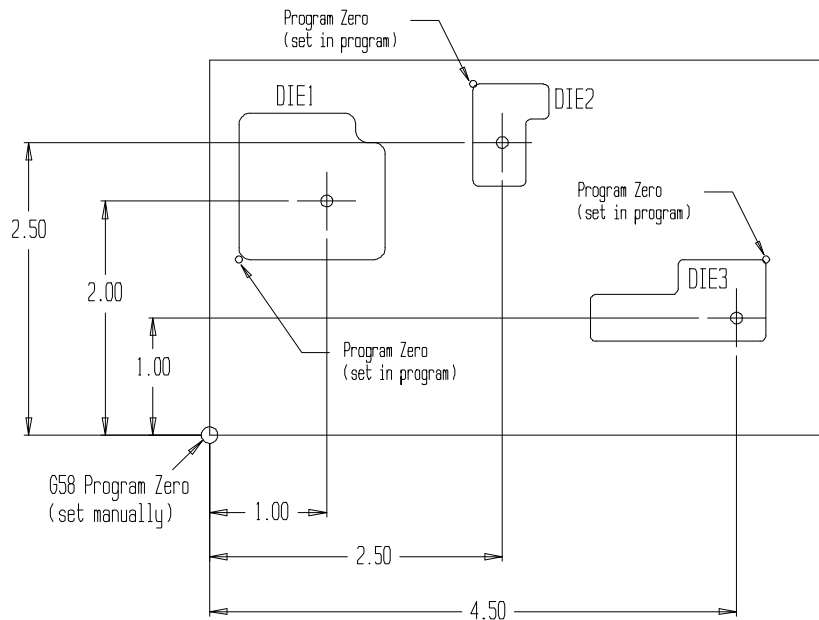


Figure 10-3

One application for coordinate system shifting is when you must cut multiple openings in a piece of material and you want the program to be automatic. Unless you have an automatic wire threader, you know that there will be some manual intervention with multiple parts (to cut wire and rethread wire for each opening). If you do have a wire threader, the techniques shown here can help, but our discussions will assume you do have a wire threader.

For programming this type of job with an AWT equipped machine, it is recommended that a CAD / CAM system be used. This is because the CAD / CAM can make the program to run unattended much easier than manually "Q"ing a series of programs.

IMPACT CAD CAM for example is a recommended software that can write the program using the following machining orders.

Rough cut all openings up to the glue tabs.

Stop the machine

Cut off all TABS, with MOO stops for the operator to remove slugs.

Skim cut all openings without stopping.

Figure 10-3 is a drawing that shows three die openings to be machined in the same piece of material.

Notice that each start hole has been dimensioned from the lower left-hand corner of the part. Using techniques you already know, you could use the Q routine 0165 to pick up that corner. If you pick up the corner of the part while you are in coordinate system number 4 (G58), you will be able to easily reference that coordinate system at any time during the program (note that it does not have to be G58, it could be any coordinate system except G54 or G959.)

Say the left die-opening NC program was named DIE1, the middle die-opening NC program was named DIE2, and the right die-opening program was named DIE3. Each of these NC programs could be modified to be a Q routine by deleting the M02's from the end of each program, then this ALLDIE program would work:

Program named ALLDIE:

N005 G58 (SELECT COORDINATE SYSTEM FROM LOWER LEFT CORNER)

N006 G92X0Y0

N010 G00 X1. Y2.(RAPID TO FIRST LOCATION)

N015 T91 (AUTOMATIC WIRE THREAD)

N016 G54

N020 QDIE1 (THIS RUNS ENTIRE DIE1 PROGRAM)

N025 T90 (CUT WIRE)

N030 G58 G00 X2.5 Y2.5 (RAPID TO SECOND LOCATION)

N035 T91 (AUTOMATIC WIRE THREAD)

N036 G54

N040 QDIE2 (THIS RUNS ENTIRE DIE2 PROGRAM)

N045 T90 (CUT WIRE)

N050 G58 G00 X4.5 Y1. (RAPID TO THIRD LOCATION)

N055 T91 (AUTOMATIC WIRE THREAD)

N056 G54

N060 QDIE3 (THIS RUNS ENTIRE DIE3 PROGRAM)

USING A COORDINATE SYSTEM TO KEEP TRACK OF THE MACHINE'S POSITION

One VERY helpful technique for Coordinate System Shifting is to keep track of the

machine's position relative to the limits of the machine. If you are going to be running a lengthy job and you are worried about the power in your shop going off, you can use this technique to assure your ability to easily get back to your program's starting position.

We have reserved coordinate system number G959 for this purpose. Every time you start the machine, you must send the machine to its negative limits (this is part of machine startup.) While the machine is at its limits, the X ,Y,U,V and Z displays for coordinate system number G959 should be zero.

Move the machine to its starting point for the program (using whatever techniques are necessary.) BEFORE you run your program, look at coordinate system number G959 and write down the X and Y values. This will be the distance from the machine's home to your start point. If the power goes off during the machining of your part, you will be able to easily send the machine back to the start point. Follow the steps below.

Power up the machine.

Go to machine home limits by doing G81X-Y-.

Switch to work-coordinate G959.

Then command a G00 in MAL, MDI back to the numbers you wrote down before.

USING COORDINATE SYSTEM SHIFTING TO COMPARE MEASUREMENTS

When it is critical that you make edge-find as perfectly as possible, it is sometimes wise to repeat the edge-find results and compare for accuracy. You can make the edge-find find in one coordinate system (G54 for example) and zero the axis display, and then switch to another coordinate system (G55 for example) and repeat the edge-find. Press the F4 key and look to see the difference in the zero position of G54 when you're done.

To compare center finds:

Type the following in MDI

G54 Q0145

G55 Q0145

And press Enter

When the center finds are done, press the F4 key to compare the difference between the G54 and G55 work coordinates.

MIRROR IMAGE AND AXIS EXCHANGE

These features allow the program to be written in the most convenient way. Then if the part is setup differently than the way it is programmed, you can rotate or mirror the program to match the way the part is setup, instead of breaking down the setup. This can be a little confusing so always dry run the program prior to actually cutting your part to be sure it will come out correctly.

All of these features can be programmed or set manually in the SET screen. However, if you set any of these features in the SET screen, they may be canceled at the completion of the program (by M02.) This means that after you do a dry run, you will have to activate them again.

For this reason, it is wise to include the proper G codes in the program. Then you will not have to worry about remembering to turn them back on after a dry run.

There are several “G” codes involved with these features:

G05 - Turn on X axis mirror image (offset & taper direction changes)
G06 - Turn on Y axis mirror image (offset & taper direction changes)
G08 - X Y axis exchange
G09 - Cancel mirror codes G05, G06, G08

When you turn on a mirror image, the direction of offset (G41 and G42) will be reversed automatically in order to machine the shape to size, although the offset register in the run screen doesn't show that.

When taper cutting, the taper direction also needs to change. Keep in mind that the machine is traveling in the reverse direction because of the mirror code. It might seem as though the taper is also mirrored automatically, but it is not. If you set your job upside-down on the worktable, but it was programmed face up, you must switch the taper code G51 to G52 in the program and table to program in the SET screen.

Figure 10-4 on the next page is a chart that shows all of the various possibilities for mirror image and axis exchange.

When you are finished using mirror image or axis exchange in a program, you **MUST** remember to cancel them (with G09.) We recommend placing the cancellation command just prior to the M02 (end of program.) If you forget to cancel these features prior to the end of a program, you will receive no alarm, but the machine will still be under the influence of the feature/s programmed, meaning your next job may be incorrectly run. The axis under the influence turns red on the CRT.

Mirror Image And Axis Exchange Chart

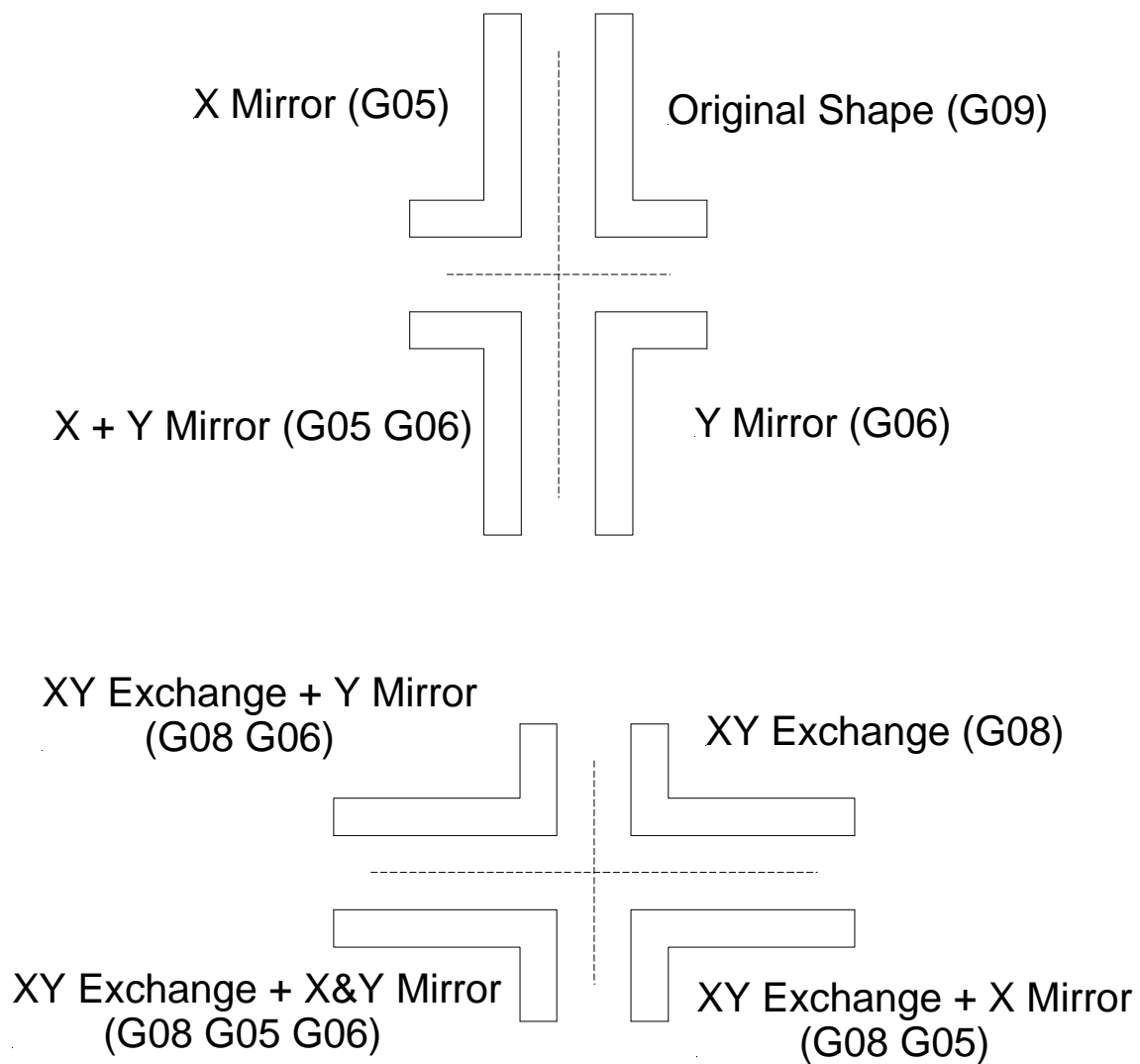


Figure 10-4

SUBROUTINE TECHNIQUES

We discussed sub-routines during our presentation of program formatting (chapter 9.) We gave the most useful reasons for using sub-routines, which are to repeat motions for trim passes in that session. So, our presentation here will be limited to the format of sub-routines.

When the control reads an M98 with a P word, it jumps to the sequence number specified by the P word. It continues execution from that point until it reads an M99 command. Then it jumps back to the command just after the M98 command. If L is included in the M98 command, the L word tells the control how many times to loop the sub-routine. If the L word is not included in the M98 command, the sub-routine will be executed once. One very good application for a sub-routine is to avoid having to re-program trim passes. This was well presented during our discussion of program formatting in Chapter 9, so we will not give an example here.

SCALING

(Input as a 3 place value ex: 1000 = 1 to 1)

Scaling allows you to easily change the size of the workpiece to be machined. With an easily input scale factor, you can modify the cutting motions to make a larger or smaller shape. However, you must be careful not to break any rules related to wire radius compensation. For example if you had an original shape that included .005 inch inside radii, and you scaled down to half size, the inside radii would now be .0025 and the original wire diameter would probably not work.

Scaling does not require any program commands. The scale factor is entered into the SET screen or in the RUN screen. 1000 is equal to full scale (note that no decimal point is allowed.) For half scale you would set the scale factor to 500, for twice the original size, you would set the scale factor to 2000 and so on. If the version of software in the control is 3.00JPY or later you can enter it into the program above the G92 statement.

NC program example “**FLAG3D2000**” scales the program up twice the size.

Once the scale factor is set, the control will modify all movements by the amount of the scale factor.

Note: scaling does not effect the offset value. Also, an icon appears in the run screen if set to a value other than 1000 which is full scale.

AUTOMATIC CORNER ROUNDING

This feature was discussed during our descriptions of motion types (concept number three about G02 and G03 circular commands.) Here we will take it a little further. However, remember that these techniques may not perform the desired taper for die machining. If you use these techniques, the radius at the top of the die will match the radius at the bottom of the die. The use of G02 and G03 with the I and J is a better method for making circular movements on a die, since the radius will grow with the taper in the die. However, if you are programming shapes that do not require conical taper or no taper at all these techniques can make manual programming much simpler.

The basic idea with automatic corner rounding is that you are allowed to program directly to the **intersection point** of two surfaces and simply include an “R” word (for the desired radius) in the command to the intersection point. The control will automatically start rounding when it approaches the intersection point.

The information on page 5-5 through 5-7 and the information on this page show you how to use it.

USING AUTOMATIC CORNER ROUNDING “R WORD”

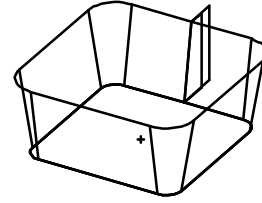
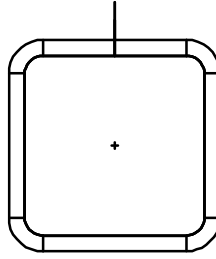
Notice with automatic corner rounding that block number N0011 included an R word to tell the control to start rounding as it approached X-.5. Also notice that the X-.5. is the intersection point to the next surface to machine. While this technique is quite simple to understand, remember the limitations related to taper cutting. You may be tempted to use this technique to program a punch (since a punch does not have any taper). But many times you will want to use the punch program to make the die program. In most cases you will have to use G02 and G03 to accomplish this.

NOTE: Offset code G41 (or G42) is required in the program for the R word to work. If you want to cut without offset, simply include the G41 (or G42) code and use H000 for the offset value.

```

"N0002 (MAIN PROGRAM);"
"N0003 G54 G90;"
"N0004 G92 X0. Y.8;"
"N0005 G29;"
"N0006 T84;"
"N0007 C511 H1=.0061;"
"N0008 G42 G52 A0 H0;"
"N0009 G01 Y.5;"
"N0010 A10.0 H1;"
"N0011 X-.4;"
"N0012 G03 X-.5 Y.4 I0. J-.1;"
"N0013 G01 Y-.4;"
"N0014 G03 X-.4 Y-.5 I.1 J0.;"
"N0015 G01 X.4;"
"N0016 G03 X.5 Y-.4 I0. J.1;"
"N0017 G01 Y.4;"
"N0018 G03 X.4 Y.5 I-.1 J0.;"
"N0019 G01 X0.;"
"N0020 A0 H0;"
"N0021 G40 G50 Y.7;"
"N0023 M02;"

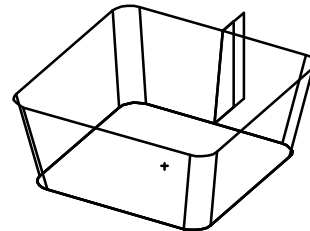
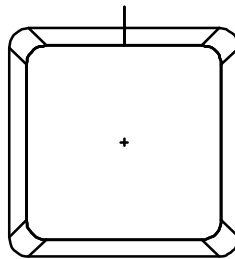
```



```

"N0002 (MAIN PROGRAM);"
"N0003 G54 G90;"
"N0004 G92 X0. Y.8;"
"N0005 G29;"
"N0006 T84;"
"N0007 C511 H1=.0061;"
"N0008 G42 G52 A0 H0;"
"N0009 G01 Y.5;"
"N0010 A10.0 H1;"
"N0011 X-.5 R.1;"
"N0012 Y-.5 R.1;"
"N0013 X.5 R.1;"
"N0014 Y.5 R.1;"
"N0015 X0.;"
"N0016 A0 H0;"
"N0017 G40 G50 Y.7;"
"N0019 M02;"

```



AXIS ROTATION

This feature allows movements to be rotated about the G92 point in the program. This feature can be used when the part is setup differently than the way it was programmed. The example below shows how to rotate the program 90 Degrees about the G92 point.

G26 RA90. (turn rotate on and rotation angle of 90 degrees)

G90 (absolute mode)

G92X0Y0 (start point location)

(the rest of the program here)

G27 (cancel rotation)

M02 (end of program)

STRING FILE

If you have other wire EDM equipment (other than Sodick), this feature will allow you to make the Sodick machine respond to commands that are used in your other machines. You can think of the string file as a translation table. It lets the Sodick machine read and execute commands that it would not normally recognize. It can also convert single codes to mean more than one code.

Other machines may use an M86 command to turn on flushing. Your Sodick machine has no use for the M86 and normally it would generate an alarm if an M86 were read. On the Sodick machine, you could use the string file to tell the machine that whenever it reads an M86, to perform the equivalent function which is T85 (flushing on for Sodick.)

There are some limitations to what the string file can do. For example, some machines utilize the I and J for circular movements in a totally different way than Sodick. On some machines, the I and J are the actual absolute positions of the center of a radius, instead of the incremental distance from the start point of the radius to the center. In this case, nothing could be done to make the program files from the other machine compatible with the Sodick control. The string file is stored on the user floppy disk or on the hard drive and it can be loaded into memory and in the edit screen, you can view and change its contents.

NOTE: The control uses the string file which is stored on the hard drive or floppy drive, not the one in memory. If you have to change the string file, change it in the edit screen, and then be sure to save the file back to the user disk or the hard drive for the changes to take effect. (See next page for an example of the format and some recommended settings).

EXAMPLE OF STRING FILE

T84:T84G104X5. *(Multiple code conversion example)*

T85:T85G105

LIMITS:G81X-Y-U-V-Z-

STOP:M00

G80:T80G80

SSP:G59G92XYUVZ

SP:G59M05G00XYUV

SUB:T88

HALF:G82

T88:T94T82T96

T94:T94T82T96

T89:T89T83T97

DRN:T83T97

FIL:T82T96

G254:G254

G255:G255

G256:G256

G257:G257

G258:G258

G259:G259

G454:G454

G455:G455

G456:G456

G457:G457

G458:G458

G459:G459

G25:G92

G43:G41

G44:G41

G45:G40

G60:G28

G96:G09

M20:T91

M21:T90

M22:M06

M30:M02

M40:

M80:T84 T86

M81:T85 T87

M82:T80

M83:T81

M84:

M85:

M90: (Continued on next page)

M91:

T01:C000 D000

T02:C001 D001

CHAPTER 11 PRACTICE EXERCISES

This set of practice exercises was designed to reinforce your understanding of the material presented during the Course.

PRACTICE EXERCISES CHAPTER ONE

Description of the EDM Process

General EDM Factors:

1) Discuss how each of the following Factors contribute to the EDM process:

A) Wire Diameter:

B) Flushing:

C) Wire Speed:

D) Wire Tension:

E) Water Resistivity:

F) Workpiece Material Hardness:

3) How do you determine the proper wire diameter to use for any one workpiece?

4) When would you want the flushing to be set to high flush?

Low Flush?

5) When would you need a faster wire speed?

Why would you consider slowing the wire speed down for finishing?

6) What will happen if the wire tension is set too low?

7) When is it not wise to use “tap water” to fill the dielectric tank on a wire EDM Machine?

8) Which kind of tool steel cuts faster, the material in its soft state or the material in its hardened state?

Parameters Specific To Sodick:

9) Name the parameters that have the tendency to “add power” to the wire during cutting.

10) Name the parameters that have the tendency to “take power away” from the wire during cutting.

11) Name the parameter (one or two letter abbreviation) that is related to each description.

A) Time that the power is removed from the wire:

B) Time that the power is applied to the wire:

C) These TWO parameters control the motion rate of the table:

D) Multiplier for the off time (part of the adaptive control):

E) Peak Current

F) Capacitance

G) Voltage

12) Explain the first digit of MA.

The second digit?

13) If the table was moving too fast, and DC arcing was occurring, what would you do to SF and/or SV to attempt to rectify the problem?

More On EDM:

14) Where do you go to find the specific Condition Numbers to machine a specific material, thickness, with a particular wire diameter and number of trim passes?

15) What programming word (letter address) specifies the condition?

PRACTICE EXERCISES CHAPTER TWO

Machine Layout Description

1) Give the axis name (letter address) for the following descriptions:

A) Table motion in and out (toward you and away from you).

B) Table motion left to right.

C) Upper guide motion left to right.

D) Upper guide motion in and out (toward you and away from you).

2) Name the button corresponding to the following descriptions.

A) Activates a program.

B) Temporarily pauses the program operation.

C) Acknowledges the fact that you have read a warning or alarm.

D) Permanently stops the activation of a program.

3) What does the “KYB / REM” switch do?

PRACTICE EXERCISES CHAPTER THREE**Program Zero Description**

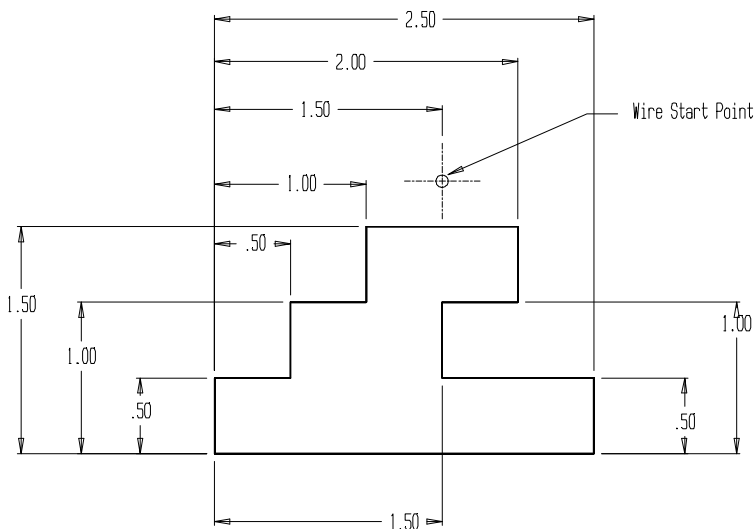
3) What is the purpose of using a program zero point?

4) How do you decide where to place program zero?

5) What G code in your program assigns the program zero point?

6) For the following drawing, mark up where program zero should be located and give a corresponding G92 command based on the current wire location:

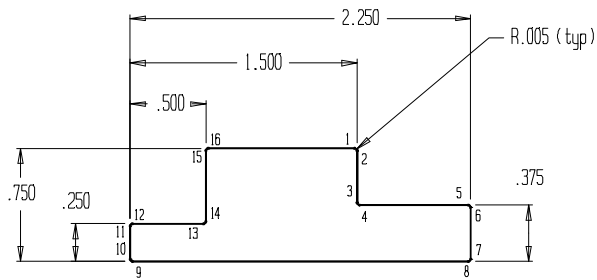
G92 X_____ Y_____



7) What do the series of G codes, G54 through G959 do?

8) For the drawing to follow, fill in the corresponding coordinate sheet. Then, mark up a location on the drawing that would be an acceptable start point for the punch program

and another for the die program. And last, tell what wire diameter would be best to machine this part.



Maximum Wire Diameter= _____

Coordinate Sheet

#	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		

Best wire start point for a die?

X _____

Y _____

Best wire start point for a punch?

X _____

Y _____

PRACTICE EXERCISES CHAPTER FOUR

Program Makeup

1) What letter signifies line numbers?

2) What are the four steps you should take before attempting to write any CNC wire EDM program manually?

- 1) _____
- 2) _____
- 3) _____
- 4) _____

3) How do you choose the correct wire size for your job?

4) How do you choose the starting point of your cut?

PRACTICE EXERCISES CHAPTER FIVE

The Four Kinds Of Wire Motion

1) What is the G code for rapid motion?

Straight line cutting motion?

Clockwise Circular motion?

Counter clockwise circular motion?

2) When would you ever consider using a rapid motion in your program?

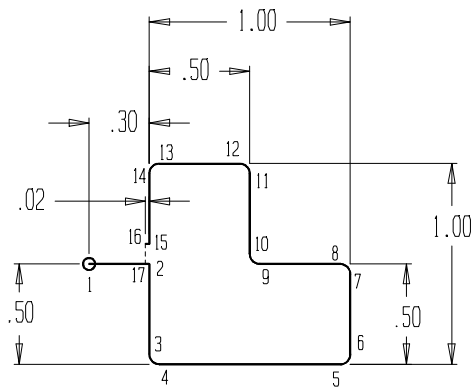
3) Name the machine related functions that will also occur when a straight line motion or circular motion command is given:"

4) What does the word MODAL mean?

5) What are the two ways to specify circular motions?

Which is the way we recommend and why?

6) For the following drawing and program (on the following page), fill in the "blanks" to complete the program.



Coordinate Sheet

#	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		

Program:

N005 G54

N010 G92 X_____ Y_____ Z_____ (1)

N015 G90

N020 C411

N025 G42 H061

N030 T84

N035 G01 X_____ (2)

N040 Y_____ (3)

N045 G03 X_____ Y_____ I.05 (4)

N050 G01 X_____ (5)

N055 G03 X_____ Y_____ J.05 (6)

N060 G01 Y_____ (7)

N065 G03 X_____ Y_____ I-.05 (8)

N070 G01 X_____ (9)

N075 G02 X_____ Y_____ J.05 (10)

N080 G01 Y_____ (11)

N085 G03 X_____ Y_____ I-.05 (12)

N090 G01 X_____ (13)

N095 G03 X_____ Y_____ J-.05 (14)

N100 G01 Y_____ (15)

N105 G40 X_____ (16)

N110 M00 (GLUE STOP)

N115 G01 Y.5 (CUT OFF)

N120 X-.3 (MOVE BACK TO START POINT)

N125 M02 (END OF PROGRAM)

PRACTICE EXERCISES CHAPTER SIX

Description Of Programming Codes

1) Give a brief description (or use) for the following “letter addresses”.

N word

G word

X and Y

U and V

T word

H word

C word

M word

R word

2) What is the “slash” code used for?

PRACTICE EXERCISES CHAPTER SEVEN**Wire Offset Description**

1) What are two reasons for using wire radius compensation?

2) What are the programming words related to wire radius compensation? What are their meanings?

3) For the following H words, give the numerical value of the offset when in 4 place inch.

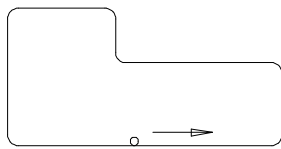
A) H055 = _____

B) H087 = _____

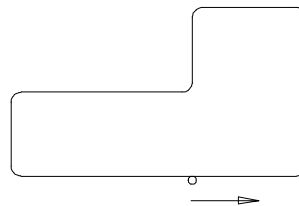
C) H065 = _____

4) What are the three steps to using wire radius compensation?

5) For the following examples, circle the correct G code.

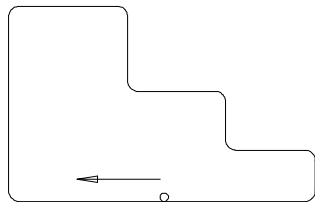


G41 or G42? _____

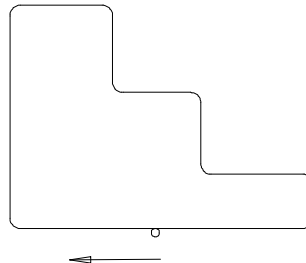


G41 or G42? _____

G41 or G42? _____



G41 or G42? _____



6) Say you had to machine a punch and die. Based on the workpiece material, wire diameter, workpiece thickness, and finish requirements, you looked in the SODICK condition manual and found that you needed three total passes (rough and two trims) to machine the parts to the desired size and finish. You want to machine the PUNCH ON SIZE and incorporate .003 inch die clearance per side in the die. Here are the “on size” offsets you found in the SODICK condition manual:

Rough pass: H087

First trim: H056

Second trim: H053

What values would you use as the offsets to make the die with .003 per side clearance?

Rough pass: H_____

First trim: H_____

Second trim: H_____

7) What is the top of a program called where cutting conditions and offsets appear?

8) What is the purpose for machining a test square?

9) Say you machined a .500 inch square test punch with two passes and .010 diameter wire. Say roughing pass used an offset of .0088 and the finish pass used an offset of .0055 inch. After cutting the test square, the finish looked OK, but the size was .5006 inch square. What would you do to the two offsets to make the next workpiece cut with the same machining conditions come out on size?

10) If making the punch “on size” and you want to incorporate .002 inch clearance per side in the die, what would you do to the “on size” offsets to machine the die with the proper clearance?

11) Why is it important to incorporate a small radius (bigger than the wire radius) on ALL OUTSIDE corners of punches and dies?

12) If machining punches and dies, is it wiser to run the punch or the die first?

Why?

13) When the required die clearance is greater than the smallest offset to be used in the program, what may be used to incorporate the die clearance?

PRACTICE EXERCISES CHAPTER EIGHT

Taper Cutting

1) What are some popular reasons for taper cutting?

2) What are the programming words related to taper cutting and what is each one's meaning?

3) What are the three steps to programming taper?

1) _____

2) _____

3) _____

4) If you were machining a die with the top of the die “up” and the program commanded that the wire move around the die in a “counter clockwise” direction, which G code would you use to instate taper?

5) Define the following parameters (in relation to the worktable):

Table To Limit :

Table To Upper :

Table To Lower :

Table To Program :

Table To Next :

PRACTICE EXERCISES CHAPTER NINE

Programming Aids*Q Routines:*

1) What is a Q routine?

2) When would you consider writing a Q routine?

3) How does a Q routine end?

4) How are variables represented in a Q routine?

5) What is the purpose of brackets in a Q routine?

6) How are variables “passed” from a call statement to the Q routine?

Coordinate System Shifting:

8) What were the three applications given for using coordinate system shifting?

9) Name the G codes related and their corresponding system numbers:

Subroutine Techniques:

10) What are the four words related to subroutines?

11) When is it most helpful to use subroutine techniques? (based on our discussions).

Automatic Corner Rounding:

12) What is the difference between using G02 and G03 as they compare to using automatic corner rounding.

13) With automatic corner rounding, what programming word (letter address) specifies the desired radius?

14) What is the major limitation to using this feature (making it almost mandatory to use G02 and G03)?

Scaling:

15) Is there any programming word related to scaling?

16) If you wanted to run a program, scaling it up to twice programmed size, what value would you enter as the scale factor (in the RUN mode)?

Mirror Image and Axis Rotation:

17) What is the difference between G05 mirror image X and G93 mirror image X? Which one is almost NEVER used.

18) What is the purpose of having these features (mirror image and axis exchange)?

Axis Rotation:

19) What are the programming words related to axis rotation?

What are their functions?

String File

20) What is the purpose of the string file?

CHAPTER 12 MAINTENANCE

Sodick Wire EDM's are precision machine tools and should be treated as such. Preventative maintenance will eliminate unnecessary problems, downtime, and aggravation and should be performed on a regular basis religiously. Sodick Wire EDM's had a reputation in the industry to run longer and in dirtier conditions between maintenance intervals than their competitors. This does not mean maintenance is not as important and should be put off until you start having problems with the machine. "Don't fix it if it ain't broke" theory doesn't apply here. Ideally, the machine should be located in an environment where the air is clean (not right next to a grinder) and temperature doesn't fluctuate more than several degrees (not located in direct sunlight or by air duct, etc.)

NOTE: To remove the EDM residue that accumulates on the machine and on the workpieces, cleaners that contain phosphoric acid, hydrogen chloride or equivalent are needed. Toilet bowl cleaners SNO-BOL and ALLOUT work well. AC-500 is a cleaner that works well and is not as harsh as toilet bowl cleaners (1-800-842-3919). These cleaners should be rinsed off within several minutes with water and will decrease the life of the DI resin if too much of it gets into the system.

DISPLAY MAINTENANCE SCREEN

In the "DSPLY" "MENTE" screen (figure 12-1), there is a bar graph that will turn red when the item is due for service. It can be reset after an item has been serviced, returning the bar graph to 100% and green. The setting for "MAX TIME" is adjustable, and should be set to a value. The value "TIME" displays the machining hours that the item has.

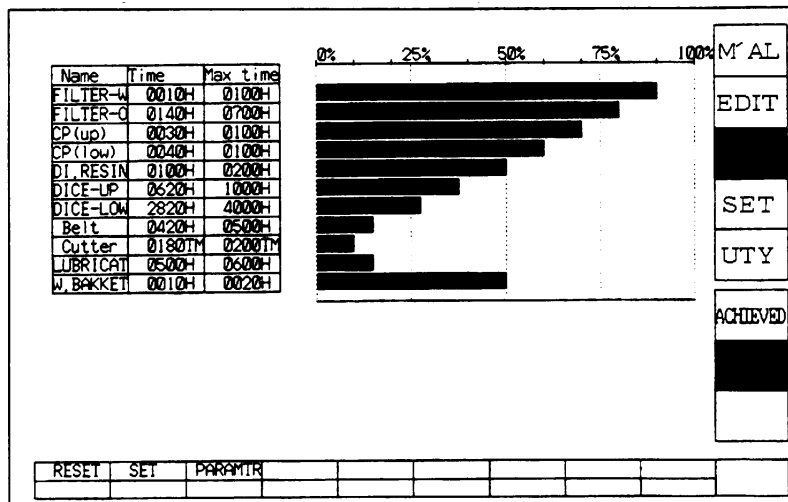


Figure 12-1

MAINTENANCE CHECKLIST

BY THE HOUR CHECKS

- 1) Carbide power feeders every 70 Hours. See page 12-4

DAILY CHECKS

- 1) Water level in the dielectric tank. See page 12-4
- 2) Water filter pressure. See page 12-4
- 3) Wipe off the grit on the gray round portion of the lower arm.
- 4) Water resistivity . See page 1-4 & 12-3
- 5) Squareness of the wire. See page 14-7
- 6) Upper and lower flushing nozzles. See page 12-4
- 7) Aspirators. Check for slurping sound at work tank front door.
- 8) Wire disposal bin . Empty if more than half full.
- 9) Lower wire roller. See page 12-5
- 10) Wire ejection rollers . See page 12-5
- 11)AWT Unit See page 12-6
- 12)Wire Cutter Unit See page 12-7

WEEKLY CHECKS

- 1) Clean air filters on control and water chiller.(blow off dust)
- 2) Clean floppy disk drives using cleaning disks.
- 3) Grease machine ways. See page 12-6
- 4) Clean and lubricate (grease film) the upper wire guide slide on machines with AWT.
- 5) Clean the worktank and table. See page 12-4

BI-WEEKLY

- 1) Clean lower head and check lower head alignment height. See page 12-10
- 2) Remove and clean lower wire roller. See page 12-5

MONTHLY

- 1) Clean resistivity probe inside the dielectric tank.
- 2) Clean discharge cable ends inside worktank. (emery cloth)
- 3) AWT machines: Drain accumulated water from the air pressure regulator.
- 4)AWT machines: Check oil level in air hydro unit for upper guide (on side of machine). Refill with AWT oil from Sodick Parts dept. or Mobil DTE 24,SIO grade 32, sold by Grainger (p.n. 4ZF33). This is a spindle oil. Both resevoirs floats should be between the lines when full.
- 5)Clean wire guides. See page 12-5

QUARTERLY

- 1) Grease machine. See page 12-6 or your “Machine Tool” manual.

ANNUALLY

Drain ,clean and refill dielectric tank.

MAINTENANCE DESCRIPTIONS

RESISTIVITY METER

This meter measures the resistance of the deionized water from 0 - 20 and is located below the chiller. The needle should read between 5 and 10 for most cutting. The red toggle switch is the on and off switch and must be on to obtain a reading. The P (power) LED light will light up when the red toggle switch is turned on. Flip the black toggle switch up and the needle will move toward zero. While holding this toggle switch up, rotate the ADJ knob to align the needle to zero. Once the needle is aligned with zero, release the toggle switch and the needle will now display the resistivity of the deionized water. The high and low knobs located directly below the meter are to set the high (10) and low (5) limits. There is also a H LED and a L LED. When the L lights up it means the needle is at a lower reading than what the low knob is set at. This would indicate that the DI resin is no longer good enough and needs to be changed. When the H lights up it means the needle is at a higher reading than what the high knob is set at. If the needle drops below the low knob setting the machine continues to cut. There is not an automatic shut down feature. If the resistivity gets extremely low (2 or lower) the machine may think the wire is short circuited. This will throw an alarm stating so and the workpiece could start pitting (secondary EDMing). So look at the meter everyday and verify the needle is in the range you have set. There is a probe submerged into the dielectric tank next to the flushing and fill pump that measures resistance between 2 prongs and relays the information to the meter. Clean these prongs once a month. See page 1-4 for more info.

DI BOTTLE:

If the DI bottle needs to be recharged,

1. Turn off the resistivity meter and disconnect the lines at the top of the bottle noting which one is "IN" and "OUT". Put the disconnected ends into the dielectric tank so water doesn't run onto the floor.
2. Unscrew the blue cap and pull the cap and the rod plugged into it out and rinse the DI resin off the screens with tap water.
3. Now pour out all the used DI resin from the bottle and rinse the bottle out. The old resin should be recycled.
4. Pour in new DI resin and fill the bottle about 7/8 of the way up.
5. Push the rod attached to the blue cap down until the cap can be threaded back on.
6. Plug the hoses back on to the bottle and turn on the resistivity meter.
7. If there is a valve on the supply hose (where the hose is connected to) it must be open to allow water to flow into the bottle. Also make sure the hoses don't get kinked, thus blocking the flow.

The meter should return to your setting value in an hour or so.

WORKTANK AND TABLE:

Use AC-500 or equivalent and spray down the tank and table and let sit for several minutes. A toilet bowl brush works well for scrubbing out the tank and a toothbrush for the smaller areas.

If a small amount of acid is used, wipe up excess with a cloth.

If a large amount of acid is used, disconnect the large gray drain hose where it attaches to the dielectric tank and put it in a bucket. When you rinse out the worktank, the water and acid will go into the bucket and not the dielectric system.

AC500 is available from Rochester Midland (800.842.3919)

Use only an Arkansas stone on the table to remove nicks and scratches. If a fine India stone is used, it will scratch the table and remove too much material.

WATER FILTERS:

The overflow in the dielectric tank should be the width of the opening and drop several inches with the worktank empty. If not add water and if it is narrower, the filters are getting clogged. Look at the pressure gage located on the top of the dielectric tank and verify that it is not in the red zone. Using a white container, catch some of the water. It should be visibly clean. If it is gray or slightly black the filters need to be replaced. Water is constantly flowing through the filters and the filters catch any particle that is bigger than what the filter is rated at. To replace the filters, turn the circulation pump off with the chrome toggle switch located to the right of the resistivity meter. Unscrew the wing nut and remove the metal cap, rubber washer and the brass washer and pull the old filter up and throw it away. Verify that the o ring is located on the flange where the bottom of the filter seats onto. CAUTION: The o-ring may stick to the bottom of the filter. The o-ring must be in place for the filters to work properly.

CARBIDE POWER CONTACTS:

The carbide power feeders deliver the electricity to the wire thus charging it with current. They need to be indexed when the witness line is about .010"-.015" deep or about every 70 or 80 hours. Sodick provides an indexing tool with the machine or a 6" scale could be used to track location of the power feeder. Loosen the set screw (3mm allen wrench) located inside the head of the cap screw that holds the short power cable to the front of the upper head and push the feeder about .040" and retighten the screw. The lower head also has the same style feeder, but the screw that holds it in place is located behind the head (5mm allen wrench). Loosen it and index the feeder and retighten the screw. Once the entire 1st surface of the feeder has lines on it, flip it over and use this entire 2nd side. It is a good idea to clean the feeders every other time they are indexed. You must keep track of which way the feeder is indexed so the wire is on a new area each time it is indexed.

FLUSH NOZZLES:

There are 3 sizes of flush nozzles.

- 0 to 10° 6mm Lower (3081674) 6mm AWT Upper (3081604)
- 11° to 20° 10mm Lower (3081675) 10mm AWT Upper (3081606)
- 21° to 30° 12mm Lower (3081867) 12mm AWT Upper (3081868)

Dry run the machine and visually verify that the wire isn't touching the nozzle. If it is, the nozzles with a larger I.D. opening should be installed and verify again that the wire isn't touching the nozzle.

WIRE GUIDES:

Sodick provides a round tool for removing the screw on style guides. The tool (die driver) has 2 lugs that fit into 2 slots ground on the guide. The hole drilled through the tool is to put an allen wrench through to loosen and tighten the guide. Check guide size and roundness with a comparator or a microscope when the accuracy of the parts your cutting deteriorates. See page 1-2 for information about split guides.

LOWER WIRE ROLLER ASSEMBLY:

There is a white ceramic roller 2.75"O.D. located below the lower head with 2 ceramic guides surrounding it. The wire travels around the roller and through the lower arm, between the ejection rollers and drops in the wire bin. This roller must spin freely and should not feel sloppy. This roller can affect how reliable the AWT unit is. Occasionally a piece of wire will get lodged between the guides and the roller. That piece will jam the roller and must be removed to allow the roller to spin freely. To remove the roller assembly, loosen and remove the 2 screws located at the top of the right side of the steel plate and pull the entire assembly toward the front of the machine and remove it. Remove the guides and remove the piece of wire that is jammed and verify that the roller spins freely. To attach the guides, use a .007" thick, .5" wide and 12" long shim stock. Insert the stock between the roller and the guides. Push the guides tight against the stock, tighten the screws to secure the guides and remove the stock. NOTE: If the small rear guide is closer to the roller than the larger guide, there will be a step that the wire will hit when it comes around the roller. Verify that the roller spins freely now. To attach the assembly to the lower arm, note the plastic tube sticking out of the lower arm. Push the assembly onto the tube with the hole in the rear ceramic guide and tighten the 2 screws that secure the steel plate. Verify that the AWT unit is reliable now.

WIRE EJECTION ROLLERS:

There is a set of wire ejection rollers located above the wire bin. These rollers pull the wire from the spool, through the machine and drop it in the wire bin. The pressure between these rollers affects how the wire lay's in the bin. When the wire run is turned on, if the wire is not laying flat in the bin, increase the pressure by turning the bolt (located directly behind the rear roller) clockwise or decrease it by counterclockwise. Wait several minutes between adjustments. This will allow you to read how the wire is

lying in the bin. CAUTION: If the pressure is decreased too much, the rollers cannot grip the wire and the wire will slip. Watch the wire spool and verify it is rotating smoothly and not jerking. Occasionally, trying a different wire may eliminate the curling problem. Sodick also offers an optional wire chopper that chops the wire into small bits and drops them into the wire bin thus eliminating wire curling in the bin.

WAY LUBRICATION:

The ways should be greased once a week with lithium grease. Sodick provides a grease gun for this purpose. Locate the stickers on the machine that show where the grease fittings are and pump the gun once for each fitting. Jog the axis it's full travel to distribute the grease. If unsure, See your "Machine Tool" manual.

AUTOMATIC WIRE THREADER:

The automatic wire threader (AWT) unit allows more unattended operation of the machine. When multiple parts or openings are to be cut, the AWT unit can cut the wire, the machine will move to another location and the AWT unit will thread the wire. This eliminates the operator having to be present. Also if the wire breaks while cutting, the machine will return to the start point location of the shape being cut, rethread the wire, return to the location where the wire broke, lower the power setting so the wire doesn't break again and resume cutting. This rethread after wire break feature can be turned on and off in the SET screen at the AWT REFERENCE parameter while cutting.

There are several items that should be maintained and adjustments that can be made to assure reliable operation of the AWT unit. See page 12-11 for the alignment procedure and pages 12-10 through 12-16 for information about parameters and adjustments that affect the AWT unit.

There is a micro switch located next to the brass unit that secures the AWT pipe that senses that the pipe has moved down. This switch signals the upper wire guide to open to allow the pipe to pass through the upper head. There are 2 pulleys with a 2 piece belt wrapped around them to pull the pipe up and down. If the belt gets kinked, it may not go around the pulleys. A backup should be kept in stock in case the belt needs to be replaced. The upper pulley is driven by an electric motor. The pipe travels down until it contacts the stop block. This block is at a fixed location on the A320D. The water jet then turns on and acts as a guide for the wire to follow. The drive rollers located above the pipe feed the wire until it reaches the ejection rollers. The pipe then retracts and the brass unit contacts the micro switch and the guide closes and the jet turns off. The AWT cycle is now completed. At the rear of the upper head is a sensor with a red LED that senses that the upper wire guide is truly closed and illuminates. The pipe can be removed by removing the round nut at the top of the pipe. When it is reinstalled, the alignment procedure on pg. 12-11 should be done or put a reference mark on the pipe and reinstall it with the mark in the same location as before it was removed thus eliminating the realignment procedure. The valve located on the front of the threader adjusts the amount of flow spraying out of the pipe. The AWT column pressure knob located below the resistivity meter affects the pressure going to the flow valve. Keep the knob all the way

clockwise for maximum pressure and open the valve 1-2 turns counterclockwise to obtain enough flow. If the wire will not enter into the brass unit that the pipe is attached to, remove the hex shaped cap located on the top of the unit and remove the preguide inside. This preguide may get plugged up with brass dust. Clean the guide, install it and install the cap and feed the wire through the threader. The optimum location for the end of the pipe when it goes down to thread the wire, is about .5" above the lower wire guide. If the pipe cannot fit into the start hole in the workpiece, set the end of the pipe about .050" above the part and tighten the stop in this location and the threading jet will shoot through the part to the lower guide. When the stop is all the way down and the pipe moves down to contact it, if the head is high up because the part is thick, the end of the pipe is several inches above the lower guide.

In this case, the worktank will need to be drained, otherwise the jet will break up as it tries to push through more than about 1" of standing water. See page 12-12 for the AWT in oil (sec) parameter explanation of how to drain the worktank for the thread cycle. There is a small knob located near the top left side of the threader that affects the closing pressure between the drive rollers. The deep slot creates less pressure and the shallow slot creates more. Use the deep slot for .006" or smaller wire, and use the shallow slot for .008" and larger wire.

WIRE CUTTER:

The wire cutter unit is located above the upper head. There are 2 carbide discs and one of them slides to shear the wire during the cut cycle. These discs last for a very long time. Thousands of cuts can be done before they need to be replaced. After the wire has been cut, verify that there is not a burr on the end of the wire. If there is, replace both discs with new ones. To replace the discs, remove the threading pipe and loosen the 2 screws on the top of the retaining plate on the cutter assembly. Carefully remove the top plate and note location of the wave washer and position of the 2 discs.

LOW AND HIGH FLUSHING KNOBS:

These knobs are located below the resistivity meter. They control the RPM of the flushing pump when in high or low flushing mode. Generally the high should be rotated so the white line is at the 5 o'clock position which gives maximum pressure. The low should be at the 10 o'clock position, which will create enough pressure for skim cutting. These knobs work in conjunction with the flow meters located at the front of the machine. The red digital readout facing the front of the machine, near the resistivity meter, shows the frequency or RPM of the flush pump when in high or low flushing mode. Use this readout to track flushing pressure since the numbers change as the knobs are rotated.

LOW set to 20Hz (should get 1 Liter / minute flow).

HIGH set to 60Hz

RST BUTTON:

The RST button is located below the resistivity meter. If the circulation pump shuts off, press the RST button to restart the pump.

WIRE GUIDE ASSEMBLY DRAWINGS

FIGURE SHOWING LOWER HEAD ASSEMBLY

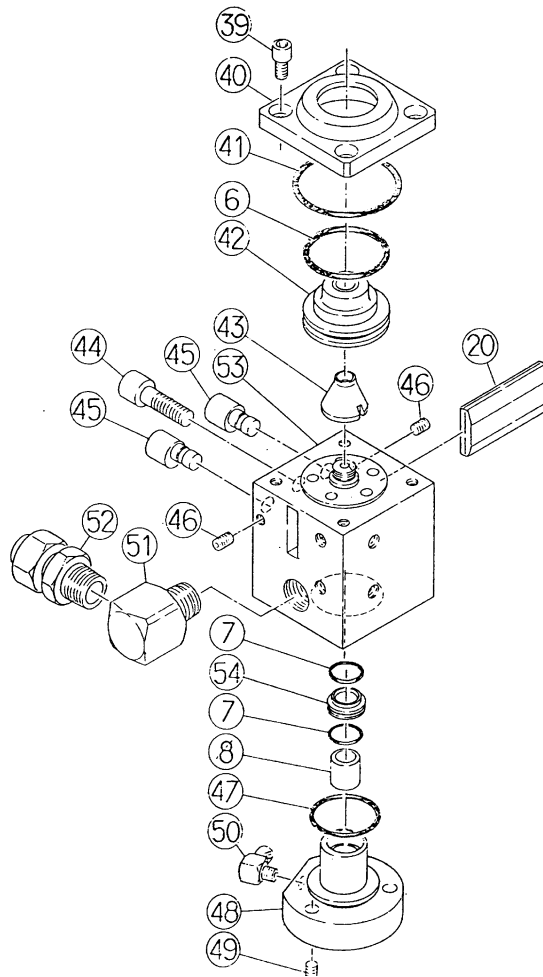
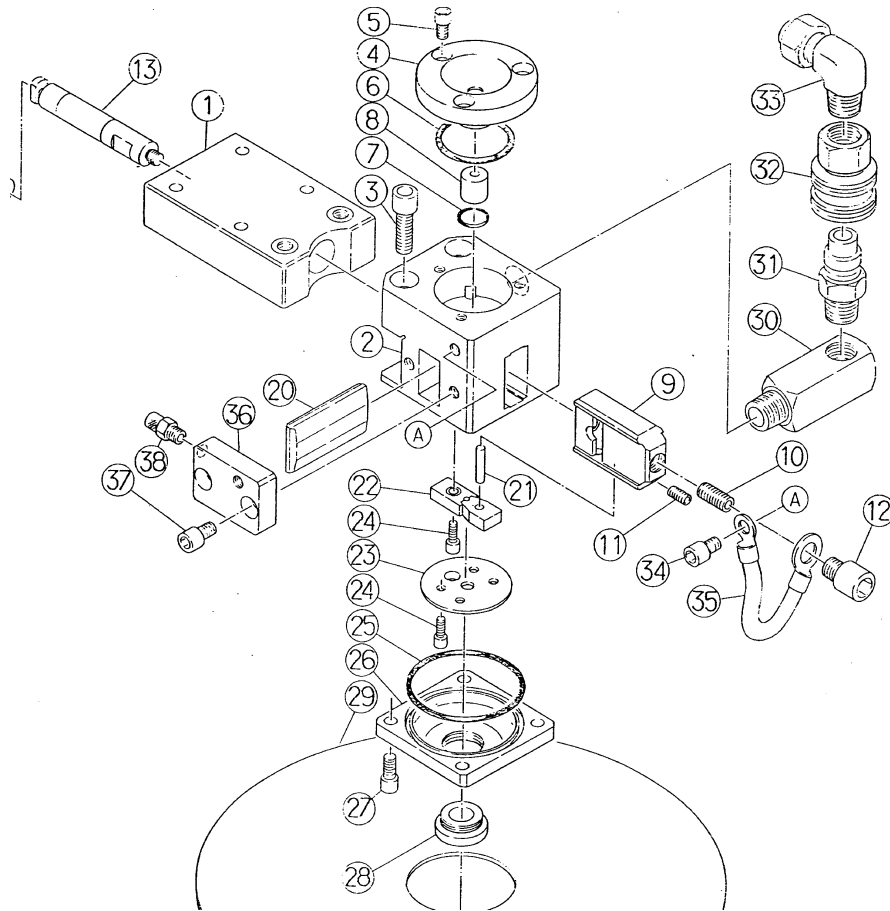


FIGURE SHOWING UPPER AWT HEAD ASSEMBLY

LOWER HEAD ALIGNMENT PROCEDURE

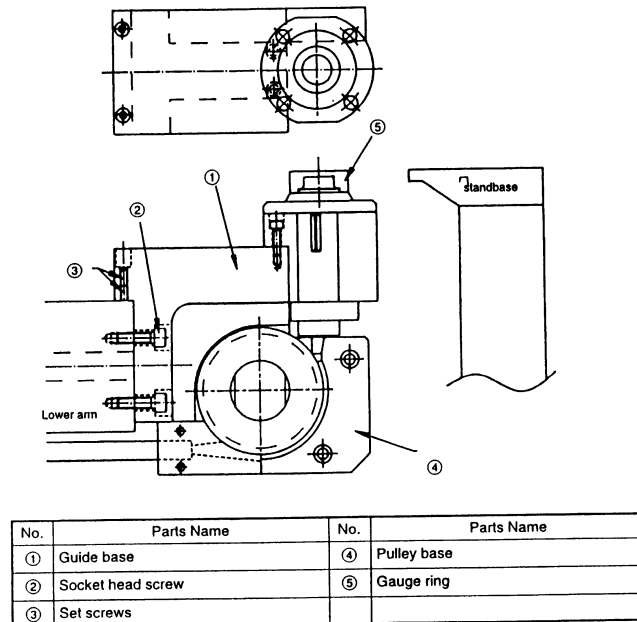


Fig. 5-17 Adjusting Lower Guide

LOWER HEAD ALIGNMENT:

The top of the lower flush cup should be about .005" below the table. To check it, take a surface gage and an indicator and zero out the dial on the table. Swing the indicator over to the inside ring of the flush cup and determine how far below the table the flush cup is. The flush cup must be held up while doing this. The thickness of a new flush cup on the inside ring is .640" and the outside ring is .628". If the reading shows the distance to be more than .005", check the thickness of the cup. If the cups are scraped or chipped or undersize, the cup should be replaced. Sodick provides a round gage that sits on top of the flange that holds the lower flush cup on. The top of the gage represents where the top of the flush cup would be.

AWT ADJUSTMENTS AND TROUBLESHOOTING

The AWT unit can be adjusted by several parameters. These parameters are located in the SET, MACHINE screen. There are also a few adjustments available in the electrical cabinet located behind the machine tool on the A320D and A530D types, and on the left side of the machine tool on A350, A500, A600, A750 and A1000 types. The wire should be square before performing the following procedure. Also, the AWT pipes have an I.D. of .7mm or 1mm and the O.D. is .079 inches. The size of the top pre-guide on the threader is .4 and the lower pre-guide should match the wire diameter.

AWT U AND V ALIGNMENT PROCEDURE

For the AWT to thread correctly, the U & V axis must move to align the water jet to the lower wire guide. This offset is calculated by the following procedure.

Remove the lower flange and flushing nozzle.

2) Set the upper head to the cutting position, then move the AWT pipe all the way down (A320D) and to about .5" above the lower wire guide on all other machines.

3) Go to "MAL", "MDI" and flip the switch on the control to W. (turns on water jet).

4) Jog the U&V axis until the jet is centralized in the funnel in the lower guide.

5) Enter the numbers in the U&V display into the "SET", "MACHINE" screen, parameter "AWT OFFSET XY (UV)".

6) Highlight "MAL", "MDI" and type UV and press enter to return the U&V axis to zero.

"SET" "MACHINE" SUBMODE SCREEN

POWER RECOVERY	= 0	WATER TO OIL TIME (SEC)	= 001	M'AL
CONTINUE DRAIN	= 1	OIL TO WATER TIME (SEC)	= 001	EDIT
AWT TYPE	= 0	FROM WATER SINK TIME	= 001	DSPLY
AWT IN OIL (SEC)	= 000	NOT USE 24	==+000000004	
AWT 1 FEED TIME (SEC)	= 000	SIGNAL TOWER	=OFF	
AWT OFFSET X(U)	==+0000.0000	NOT USE 26	==+000000000	
AWT OFFSET Y(V)	==+0000.0000	NOT USE 27	==+000000000	
AWT 2 FEED SPEED	= 000	NOT USE 28	==+000000000	
AWT 2 FEED TIME (SEC)	= 000	NOT USE 29	==+000000000	
AWT 2 UP	==+0000.0000	NOT USE 30	==+000000000	
AWT RETRY	= 000	RESERVE 31 (M)	==+000000000	UTY
WIRE BROKEN RETRY	= 000	MACHINE LOCK X	=OFF	
WIRE REMOVE RETRY	= 000	Y	=OFF	CHECK
NOT USE 14	==+000000000	Z	=OFF	
BUCKLING TIME (SEC)	= 00	U	=OFF	DISK
RESERVE 16 (P)	==+0000.0000	V	=OFF	
W.B.A.S IGNORE	=OFF	W	=ON	DATA
RESERVE 18 (M)	==+000000000	RESERVE 38 (P)	==+000000000	
RESERVE 19 (P)	=OFF	RESERVE 39 (P)	==+000000000	
ENT KEY INTERVAL TIME	= 00	HARD DISK RETRACT (SEC)	= 001	
Parameters scroll by pressing ROLL UP / ROLL DOWN.				
DISCHARGE TRAVEL MOTOR	AXIS	SIR/PTH COMP.	CUTTING PARAMETER	ON/OFF

Fig. 8-3-1 SET · MACHINE Submode Screen (a)

Figure 12-3 AWT Parameters screen

AWT TYPE This parameter is used to select the type of AWT unit to be used. Normally set to 4

- 0: No AWT unit to be used.
- 1 : Not defined.
- 2 : Not defined.
- 3 : Not defined.
- 4 : All AWT types.
- S : AWT designed to special specification.

AWT IN OIL (SEC) When the machining is to be performed in submerged mode, this parameter is used to select whether or not to drain the work tank during AWT 2 cycle. On an A320D set this to 30. On A530D's it is only needed with parts 5.0" or thicker.

- 0: Do not drain worktank.
- 1 to 999: To execute AWT operation with work tank drained .Time is specified in seconds and is entered as the amount of time it takes to drain the worktank to the lower flush cup.

AWT 1 FEED TIME (SEC) This parameter is used to set the time in seconds, for which to pull out the wire cut by AWT 1 operation. Normally set to 1

AWT OFFSET XY (UV) These parameters are used to set the distance by which to offset the U/V axes for the AWT thread cycle. This offset aims the water jet at the lower guide. This needs to be set correctly for the AWT to work.

AWT 2 FEED SPEED This parameter is used to set the speed at which the pinch rollers pull the wire for AWT2 operation. Normally set to 6.

- 0: To cause the wire pull speed to be set according to the setting of the wire speed knob.
- 1 to 12: To cause the wire speed to be controlled on the NC side instead of the wire speed knob. The higher the setting, the faster the ejection rollers will pull the wire during the AWT cycle.

AWT 2 FEED TIME (SEC) This parameter is used to set the time in seconds, for which the pinch rollers on the AWT unit feed the wire, after the pipe is down, during the thread operation. Normally set to 30

AWT Z UP This parameter is used to set the distance that the Z axis will raise if the wire breaks. If used, the Z axis raises after the wire breaks, but before the X and Y move back to the AWT rethread point. This may be used if the upper head has to clear a clamp as it travels back to the rethread point. If it is needed, enter the amount in inches (or mm) that you need the Z axis to move up. Normally set to 0

AWT RETRY This parameter is used to set the number of times by which to allow

AWT 2 (thread wire) to fail for each AWT operation. The parameter is to be set in the range of 0 to 255. Normally set to 4

WIRE BROKEN RETRY This parameter is used to set the number of times by which to allow the wire to be broken during cutting for each shape in the program. The parameter is to be set in the range of 0 to 255. If the wire is broken more times than set by this parameter, the system will stop its operation, waiting for the “RST” key to be pressed. Each time the wire breaks it raises the “A” of the “MA” parameter one number.

WIRE REMOVE RETRY This parameter is used to specify whether the system is provided with a wire tip removal device and to set the number of times by which to execute wire tip removal after each wire break operation. Normally set to 1.

0: No device provided for wire removal or not to execute wire removal operation.
1 to 999: To execute wire tip removal by number of times set for each AWT operation.

BUCKLING TIME (SEC) This parameter is used to set the time duration in seconds, for which to feed the wire backwards if the wire buckles during an AWT attempt. Normally set to 5

W.B.A.S IGNORE This parameter is used to select whether to relax the machining condition automatically during AWT operation. This function is only available for the P type control.

WATER TO OIL TIME (SEC) This parameter is used to set the time duration in seconds, for which to drain the dielectric when switching from water to oil. Used on machines equipped with both a water and oil dielectric system (not used on A320, A530, A350, A500).

ELECTRICAL CABINET SETTINGS A320D,A530D

There are 4 settings available in the electrical cabinet located behind the machine tool. These affect how the wire spool shaft, and AWT operate.

Wire Rewind: Set between 70 and 80

When the wire breaks, the AWT unit must rewind the broken wire above the upper guide. A larger wire spool requires it to be set to a higher number.

Wire Bobbin Brake: Set between 30 and 40

This is the reversing torque on the wire spool. This reversing keeps the wire from unspooling itself if the wire is stopped or broken. A larger wire spool requires a higher setting.

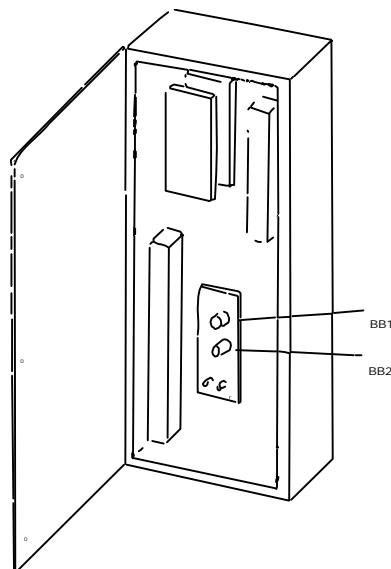
Speed control Sensing VR Knob : Set to 30.

This sets the sensitivity of the buckling sensor on the AWT unit. A lower setting makes it more sensitive to the wire buckling. Set it lower if the AWT drive rollers don't rewind the wire if it buckles between the drive rollers and the water jet pipe.

Speed Control Unit: Set to 11 or 12 o'clock position.

This orange knob controls the speed of the drive rollers and the AWT pipe speed on the AWT unit. Setting it further clockwise makes the drive rollers feed the wire faster when it is threading the wire. This unit is not installed on A320 machines.

If you adjust the speed of this unit, also adjust the parameter AWT 2 FEED TIME (SEC)



WIRE SPOOL SETTINGS A350WP,A500WP, A600WP

There are 3 settings available below the wire spool. These affect how the wire spool shaft, and AWT operate.

BB1 (Wire Rewind): Set between 70 and 80

When the wire breaks, the AWT unit must rewind the broken wire above the upper guide. A larger wire spool requires it to be set to a higher number.

BB2 (Wire Bobbin Brake): Set between 30 and 40

This is the reversing torque on the wire spool. This reversing keeps the wire from unspooling itself if the wire is stopped or broken. A larger wire spool requires a higher setting.

Normal/ Thin Wire:

This switch changes the range of wire tension. Flip to thin wire when using .004" Dia. or less wire.

Normal setting tension range = 600 to 2500 grams. Knob setting 8 will = 1200 grams

Thin setting tension range = 400 to 800 grams. Knob setting 8 will = 550 grams

AWT ELECTRICAL CABINET SETTINGS 350,500 ,600**TEN - VR:**

Used to calibrate the wire tension settings.

Sensing Adjust Knob: Set to 30.

This sets the sensitivity of the buckling sensor on the AWT unit. A lower setting makes it more sensitive to the wire buckling. Set it lower if the AWT drive rollers don't rewind the wire if it buckles between the drive rollers and the water jet pipe.

BB3:

Sets the reverse tension on the wire spool when jogging the Z axis up and down. The back tension sw toggle switch turns the BB3 function on and off.

Speed Control Unit: Set to 11 or 12 o'clock position.

This controls the speed of the drive rollers and the AWT pipe speed on the AWT unit. Setting it further clockwise makes the drive rollers feed the wire faster when it is threading the wire. This unit is recognizable by its orange knob.

If you adjust the speed of this unit, adjust the parameter AWT 2 FEED TIME (SEC)

CHAPTER 13 SET UP PROCEDURE CHECK LIST

- 1) Power up the machine. (see page 14-1)
- 2) Re-home the machine. (see page 14-1)
- 3) System checks:
 - a) Water resistivity correct? See page 1-4 and 12-3
 - b) Correct wire diameter?
 - c) Enough wire on spool?
 - d) Dielectric tank full? See page 12-4
 - e) Water filter pressure? See page 12-4
 - f) Carbide power contacts? See page 12-4
 - g) Vertically align the wire. See page 14-7.
- 4) Load your program into memory (See page 14-13 or 14-14).
- 5) Draw and Save Graphics (See page 14-21 and 15-7 thru 15-10).
- 6) Setup workpiece and indicate it straight and level (DONT SCRATCH TABLE).
 - a) Sequence to use MDI to pick up center of a hole. (See page 14-9).
 - b) Sequence to use MDI to pick up a corner (See page 14-10).
 - c) Move to the start point and set Z axis upper head location (.005" clearance above workpiece) if possible using a feeler guage.
- 7) Go to the DSPLY screen and write down machine position.
X = xxxxxxxx
Y = xxxxxxxx
- 8) Set the following items
 - Wire speed & Tension (If not in the cutting condition)
 - Flushing valves open
 - Close worktank door
 - Adjust float level for correct submerged tank depth.
 - Adjust AWT Stop for thickness of workpiece
- 9) Dry-Run the program, see page 8-10 for information:
 - a) Press the bottom vertical soft-key to go to the "RUN" screen.
 - b) Press "Arrow Up" then "Home Clear" keys to start at the top of the file.
 - c) Change the "DRY" horizontal soft-key to "1 or 2".
 - d) Press the "Enter" key.
 - e) Check for crash zones, axis limits, proper conditions & offsets. (Continued)

Note: (A320D & A530D only) If the plotting table is up and the side door is shut, DO NOT put the plotting pen arm on or it will hit the cover!!!!!!

- 10) Move machine back to the start point location and check machine numbers.
 - a) Use the horizontal soft-key “G30 START” to move back to the position set with the “G92” code.
- 11) Steps to actually start up the program. If everything looks correct, then cut the part.
 - a) Be sure you are already in the “RUN” screen from step 10 above.
 - b) Press “Arrow Up” then “Home Clear” keys to start at the top of the file.
 - c) Change the “DRY” horizontal soft-key to “0”. Do not use the “Enter” key while changing this parameter.
 - d) Press the “Enter” key to cut the program.
 - e) When machining begins, the cutting conditions can be modified to speed up or slow down the machine. See page 1-12 for tips on modifying the conditions on the rough cut.

CHAPTER 14 QUICK REFERENCE SECTION

The following section will give you step by step instruction for the following common operations.

POWER UP THE MACHINE

- 1) Turn on the main circuit breaker on the side of the control.
- 2) Place the "SYSTEM" disk into Drive 1.
- 3) Place the "USER" disk into Drive 2. (steps 2 & 3 are not done on machines with a hard drive)
- 4) Press the source button and wait until the main menu comes up on the CRT. Without hard drive it takes about 5 minutes (1 min. with hard drive).
- 5) Press the power button (to activate machine) E stop button must be popped out.

POWER DOWN THE MACHINE

All programs in memory will be lost, so be sure to save the programs.

- 1) Press the E STOP button and wait 5 seconds.
- 2) Press the SOURCE button and the light goes off.
- 3) Turn the main circuit breaker off.

REHOME THE MACHINE

- 1) After powering on the machine, check the machine home limits by highlighting "MAL", "MDI"
- 2) Type "G81 X- Y- U- V- Z-" and press "ENTER".
Machine should travel to the lower left corner, then stop.
- 3) Go to the "DSPLY" screen and the "Machine" numbers XYUVZ should be zero.
- 4) Go to "MAL", "MDI" and type UV and press "ENTER" to return UV back to zero.

Manual Z axis machines stop here (power Z continue)

- 5) If the machine has a power Z axis, Type G81Z and press ""ENTER"".
- 6) When the head stops moving go to the "SET" screen and look at the machine's paperwork to verify that "Table to Limit" and "Table to Lower" are correct. DO NOT jog the Z- axis yet.
- 7) Go to the "SET", "TRAVEL" screen and turn on "AUTO T UPPER". Now it is OK to jog the Z-axis. This parameter allows the machine to track the location of the upper head as it is jogged, and automatically set the "Table To Upper" parameter.

MOVE AXIS BY “MDI”

The Sodick control allows you to move any axis (one or more) by “MDI” in a similar way to how it is done in a program. Any rules you know related to programming will still apply to

“MDI” axis motion (G54-G959, G00, G01, G02, G03, G90, G91, etc.)

- 1) Go to the “MAL”, “MDI” screen .
- 2) If you intend to actually cut something during this motion, type “C” and the condition number you wish to use (example: C411) and press “ENTER”.
- 3) Type the mode of motion you wish (G90 or G91) and press “ENTER”.
- 4) Type the kind of motion you wish to make (G00 for rapid or G01 for straight line cutting). Note: the G01 will automatically activate electricity to the wire, turn on the wire run and flushing. Also, the current condition will be in effect for cutting.
- 5) Type the axis and motion you wish to make (example: G00 X1.5).
- 6) Press “ENTER” (motion takes place).

Notes:

- a) Machine assumes plus (+) unless minus (-) is input for axis motion.
 - b) In incremental mode (G91), movement is taken from current position.
 - c) In absolute mode (G90), movement is taken from program zero and assumes a previous G92 setting of the axis displays was done.
- Use G00 for non-cutting movements and G01 for cutting movements.

INSTALL WIRE ON THE MACHINE

There are several procedures used to install a spool of wire, based on the machine model. All machines require that you manually feed the wire all the way through the machine until it reaches the pinch rollers just above the wire bin. No matter what style of machine your company owns, there are some common notes that will apply to installing the wire.

NOTES ABOUT INSTALLING WIRE:

- 1) After you remove the wrapping from a new spool of wire mount it onto the spool shaft. Check that the wire unspools smoothly by manually pulling several yards of wire off the new spool. After the wire is wrapped around the rubber wire tension roller and the plastic rollers, it is wise to check that the wire is “pointed”. If there is any burr on the wire, you will not be able to pass it through the AWT unit or the upper and lower wire guides. One very common technique to “point” the wire is to pull the wire apart after heating it with a lighter. As the wire is pulled apart, it forms a perfect sharp point that will pass easily

through the AWT unit and the upper and lower wire guides. If the machine has an automatic wire threader, flip the switch on the control panel to cut the wire.

2) Sometimes it is helpful to release the wire tension with the wire tension knob while installing a new spool to make rotation of the wire tension roller easier. If you do this, make sure to readjust the wire tension when you are finished.

3) Be sure that the wire passes correctly through the Wire Broken sensor switch. If it does not, you will get a false wire broken alarm when you attempt to start the machining cycle.

AWT SPECIAL NOTE

4) If you have an AWT, after the wire is cut (T90 or by switch) the pinch rollers on the AWT stay clamped on the wire. To thread the wire manually, you need to press the “AWTII” switch, then the “OFF” key to open these pinch rollers.

Please remember, when changing wire spools, do not use the “AWTI” switch to cut the wire. If you do, the pinch rollers on the awt will stay clamped and you will have to do an “AWTII” to release them.

MANUALLY ADJUST FLUSHING

There are 2 flow meters on the front of the machine. These affect and show the amount of flushing flow going to the flush nozzles. There are also high and low flushing knobs located below the resistivity meter. These control the speed of the flush pump when in high or low flushing mode. There is a digital readout around the front corner showing the frequency or RPM of the pump when in high or low flushing mode.

If the high flushing knob is set too low, the worktank fills up slowly.

If the low flushing knob is set too low, you might get H161 alarm. To eliminate this alarm from appearing, adjust the low flushing knob while the low flushing is turned on so the digital readout is between 12.0 and 15.0.

Set Low Flushing To 15Hz

Set High Flushing To 60Hz

EDGE FIND BY USING THE ST KEY

This is 1 of 2 procedures used to locate the edge of a workpiece. It leaves the wire flush with the edge of the part. However, this procedure is not the most accurate. Remember

that there is a more accurate sensor touch routine that actually touches the edge three times and takes the average of the three touches.

Using the X and Y jog keys, position the wire close to the edge to be touched.

- 2) Turn wire run on manually with speed at 2 and tension at 8.
- 3) Press the (red) ST key and hold it while pressing one of the X or Y keys to jog the wire into the edge using MFR speed 1 or 2.
- 4) When the wire touches the part, it will automatically move back and stop. When finished, the buzzer will sound.
- 5) Press the "ACK" key.
- 6) The edge of the wire is now flush with the edge of the part. Press down the wire run switch to turn the wire run off.

EDGE FIND (USING G80)

The G80 code will actually edgefind and touch the workpiece three times and take the average of the three touches, making the “G80” much more accurate than the sensor touch key procedure. Set WS at 2 and WT at 8 when using .010 diameter wire.

- 1) Manually jog the wire close to the surface to edgefind (within about .100 inch).
- 2) Go to the “MAL”, “MDI” screen.
- 3) Type G80, the axis name and the direction you need to move to edgefind the part.
(choose from X, X-, Y, Y-)
(example: G80X will move the wire to the right to edgefind)
- 4) Press “ENTER”
- 5) When the edge find stops, the wire will be **flush** with the edge of the workpiece.
- 6) Set location by typing the G92 code.

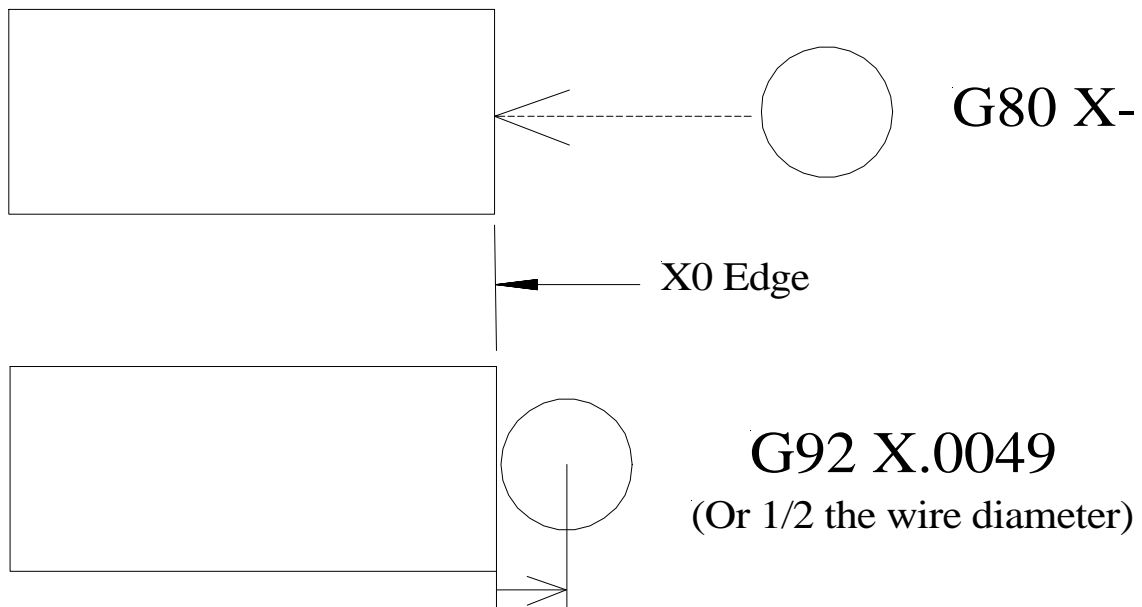
Remember 1/2 the wire size.

Always set G92 in the **opposite direction** of the G80 code.

If you used G80 X-then use G92X+.0049 (or 1/2 the wire size).

NOTE: Be sure the wire run turns on. If it doesn't, enter G80:T80G80 in the string file, or turn the wire run on manually.

NOTE: Wire run will not turn off automatically when the cycle is done.



CHANGING THE AXIS DISPLAY READ ANY NUMBER

This procedure will allow you to change the axis display to a value. This is commonly used when setting the origin.

1) Go to the “MAL”, ”MDI” screen.

To set axis display toZero:

2) Type G92XYUVZ and press “ENTER” .

To set to any number:

3) Type G92 and the values you want for X, Y, U, V, and press “ENTER” .

“G92X.500 Z0”

Note that all axis (X,Y, U,V,Z.) need not be included. If you are only interested in setting one axis

(X for example), you would type “G92X” and press enter.

VERTICALLY ALIGN THE WIRE

The vertical alignment is extremely important! If the wire is not vertically aligned, the workpiece will come out with undesirable taper due to the out of alignment condition. We recommend that you use this procedure often to test for vertical alignment.

NOTE: Before starting this procedure, be sure that the upper guide, table and the test block are completely dry. If necessary remove the flushing hose and blow air through the upper head assembly. Using a hard Arkansas stone, de-burr the test block and the table. The table is stainless steel and nicks and scratches can form. Set wire speed to 2 and tension to what will be used when you skim cut. Also set **dry run to 0**

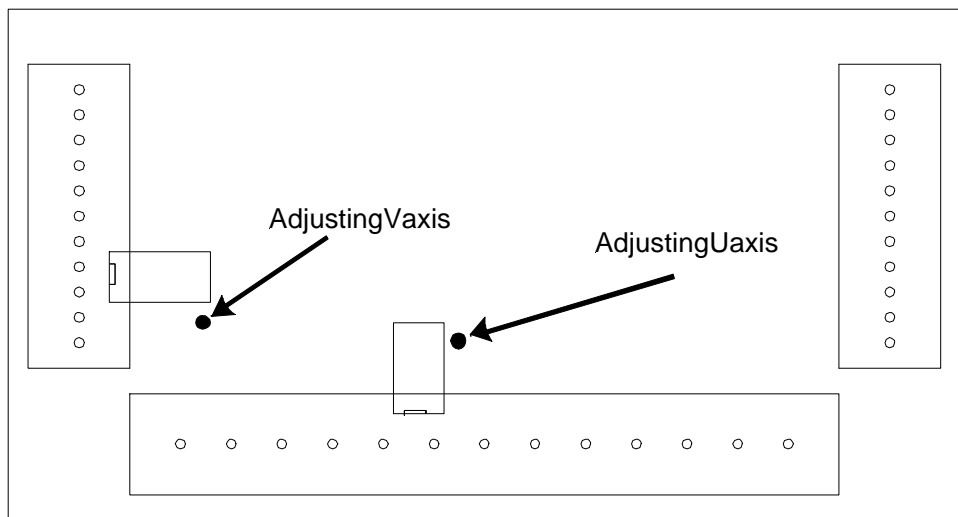
- 1) Go to the "MAL","MDI" screen, type G91 and press "ENTER".
- 2) Place the test block (that came with the machine) on the front table as straight as possible and make sure it is down good by sliding it a bit.
- 3) Type C777 and press "ENTER". (C777 is a very weak condition that will let you see the sparks between the wire and the test block).

To adjust the "U" direction:

- 4) Using the X and Y jog keys, position the wire close to the right of the test block and jog Z-axis about .500" above the test block.
- 5) Type G1X-1.5 and press "ENTER". The wire run will activate and begin moving the to the left (toward the test block).
- 6) When the wire approaches the workpiece, you should see the sparks. If you don't, there is still water on the areas that should be dry.
Use the U+ and U- keys to form a uniform spark from the top of the test block to the bottom. Note the blue "U/V key on the keyboard. You will have to hold it while pressing the U+ or U- keys.
- 7) Move the test block away from the wire (by hand) and confirm the spark again.
- 8) When satisfied, press the "OFF" and "ACK" key

To adjust the "V" direction

- 10) Reposition the squaring block onto the left or right table.
- 11) Jog X and Y to position the wire below the squaring block.
- 12) Type G1Y1.5 and press enter (the wire will activate and begin moving up).
- 13) When the wire approaches the block and begins to spark, adjust the V axis by using the V+ or V- jog keys to make the spark uniform from the top of the block to the bottom.
- 14) Move the test block away from the wire, and confirm the spark again.
- 15) When satisfied, press the "OFF" and "ACK" key.
- 16) Type G90 to go to absolute mode.
- 17) Type G97UV and press "ENTER" to zero out the U and V readout in all screens.



CENTERFIND OF A HOLE OR SHAPE

There are Q Assist routines named 0145, 0146 and 0155 designed by Sodick to make operation easier. Use these programs to pickup the center of a hole or shape. Be sure DRY = 0, WS=2 and WT=8 and the pickup surface is burr-free, clean and dry and scale is 1000

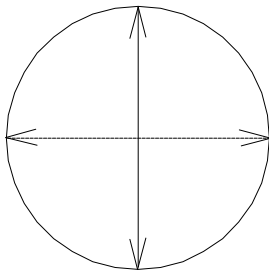
- 1) Thread the wire through the opening and confirm that it is not touching the workpiece.
- 2) Go to the "MAL", "MDI" screen .
- 3) Type Q0145 or Q0146 or Q0155 and press "ENTER"
 - a) 0146 and 0155 programs will prompt you to input a value. "ENTER" a value that is smaller than the radius of the hole.
- 4) When the routine is finished the wire will be positioned perfectly in the center of the opening.
- 5) Press the "ACK" key.

NOTE:

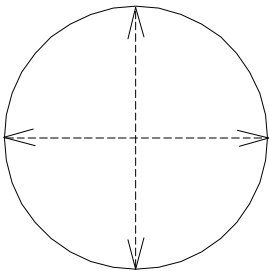
Q0145(2.0,4.0) will set the axis display to X2.0000 Y4.0000.

G26 cannot be used to rotate these programs.

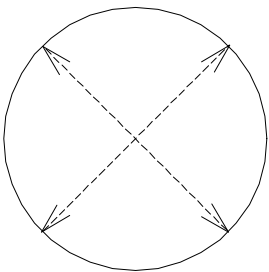
These programs can also be run from memory instead of using the Q in "MAL" "MDI".



0145.nc Program
Standard centerfind program



0146.nc Program
Centerfind program with rapid moves
Use this if pickup
hole is .500" dia.
or larger



0155.nc Program
Centerfind at a 45 degree. Also
allows input for a rapid move.

CORNER FIND ROUTINE

This is another Q Assist routine (0165) designed by Sodick to make operation easier. When you are trying to work form a corner of your workpiece (as your program zero point, for example), this routine will automatically pick up both edges of the corner and make the X and Y axis position values of that corner be any numbers you want.

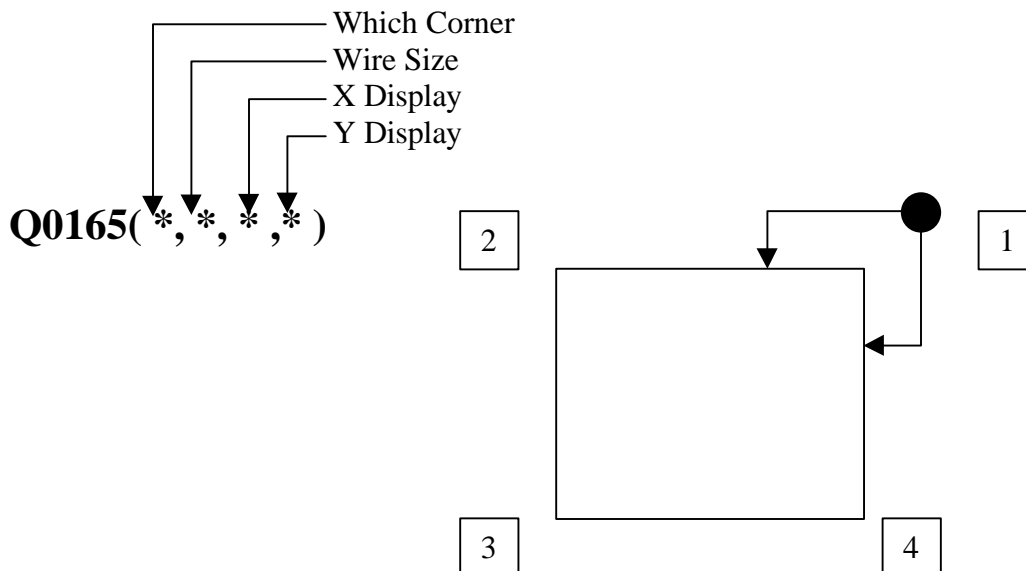
This procedure requires that you know a code number that corresponds to the corner of the workpiece you wish to pick up. The upper right corner (X+ and Y+) is corner number one. The upper left corner is corner number two, the lower left corner is corner number three, and the lower right corner is corner number four.

At the completion of this routine the center of the wire will be .050" off the corner that was picked up, and the X and Y axis displays will read the position RELATIVE to the corner.

1) Manually bring the wire to within about .100 inch of the corner you wish to pick up (in both X and Y).

2) Go to the "MAL", "MDI" screen.

Typing Q0165(1,.01,0,0) and pressing enter, picks up corner 1 using .010 wire and the corner is zero , zero on your print. You could also go to the "EDIT" screen and fill in the questions at the top of the program and then go to the RUN screen and press "ENTER".



CLEARING THE MEMORY

The following procedure clears all NC programs from memory.

- 1) Press the soft key next to ““EDIT””.
- 2) Press the soft key next to “FILE”.
- 3) Press the “Memory Clear” soft key.
- 4) Press ““ENTER”” , to delete all programs from memory.
- 5) Will you clear memory prompt appears.
- 6) Press “ENTER” for yes and “ESC” for no.

CHANGING THE FILENAME

The following procedure changes the filename o the NC file stored on the disk.

- 1) Go to the ““EDIT””, “FILE” screen and highlight “*chngrname*” with the softkey.
- 2) Move the cursor to the file you want to rename and press “ENTER”.
- 3) Input filename prompt appears in the lower left corner.
- 4) As you type the new name, it appears in the upper left corner. Now press “ENTER”.
- 5) Will you change the name prompt appears in the lower left corner.
- 6) Press “ENTER” for yes and “ESC” for No.

TYPING NEW PROGRAMS INTO MEMORY

If you wish to write and run your own manually written programs, you can follow this procedure to enter them into memory. When you are finished, don't forget to SAVE the program to the disk or hard drive to avoid losing your program when the source is turned off.

- 1) Press the soft key next to ““EDIT””.
- 2) Using the arrow keys, bring the cursor to the end of the header information in NEWFILE and press the soft key under “INSERT”. Make sure insert is highlighted.
- 3) “ENTER” the data and press the “Return” key to get to the next line.

SAVING PROGRAMS FROM MEMORY TO USER DISK (WITHOUT HARD DRIVE)

- 1) Press the soft key next to ““EDIT””.
- 2) Press the soft key next to “FILE”.
- 3) Make sure the Load/Save key is highlighting the word Save in blue.
- 4) Using the arrow keys bring the cursor to the filename you want to save.
- 5) Press “ENTER”.

- 6) It will prompt you in the lower left corner of the screen, input filename.
- 7) If the name of the file is Ok, Press "ENTER".
- 8) If the name of the file is going to be changed, type in the new name and the new name you type appears in the upper right corner of the screen. Now press "ENTER".
- 9) It will prompt you in the lower left corner, will you save?
- 10) Press "ENTER" for yes and "ESC" for no.
- 11) If you pressed "ENTER", the file is now saved to your user disk .

SAVING PROGRAMS FROM MEMORY TO USER DISK (WITH HARD DRIVE)

- 1) Insert user disk in drive 1.
- 2) Press the soft key next to "EDIT".
- 3) Press the soft key next to "FILE".
- 4) Cursor up to the word filename in the upper left corner of the screen by pressing the up arrow key on the keyboard.
- 5) Type C: and press "ENTER" (C and the colon key).
- 6) Make sure the load/save button is highlighting the word save.
- 7) Using the arrow keys, bring the cursor to the file you wish to save and press "ENTER".
- 8) It will prompt you in the lower left corner of the screen, input filename.
- 9) If the name of the file is ok, press "ENTER".
- 10) If the name of the file is going to be changed, type in the new filename and the new name you typed appears in the upper right corner of the screen. Press "ENTER".
- 11) It will prompt you in lower left corner of screen, will you save?
- 12) Press "ENTER" for yes and "ESC" for no.
- 13) If you pressed "ENTER", the file is now saved for your user disk.

EDITING USER DISK OR HARD DRIVE DIRECTLY

This function is also called "EDIT" DISK.

- 1) Go to the "EDIT", "FILE" screen and press the soft key under "Edit Disk".
- 2) Move the cursor to the file to be edited and press "ENTER".
- 3) The program appears on the screen, so you can make the changes .
- 4) To save the changes leave this screen, but when you try it prompts you in the lower Left corner, will you reflect changing on disk?
- 5) Press "ENTER" for yes and "ESC" for no.
- 6) If you pressed "ENTER" and the control has a hard drive, the changes are saved onto the hard drive. If there is no hard drive, they are saved to the user disk.

SAVING PROGRAMS FROM MEMORY TO HARD DRIVE

- 1) Press soft key next to ““EDIT””.
- 2) Press soft key next to “FILE”.
- 3) Make sure the Load/Save button is highlighting the word Save in blue.
- 4) Using the arrow keys, position the cursor to the file you wish to save.
- 5) Press “ENTER”.
- 6) It will prompt you in the lower left corner of the screen, input filename.
- 7) If the name of the file is ok, press “ENTER”.
- 8) If the name of the file is going to be changed, type in the new filename and the new name you type appears in the upper right corner of the screen. Press “ENTER”.
- 9) It will prompt you in the lower left corner of the screen, will you save?
- 10) Press “ENTER” for yes and “ESC” for no.
- 11) If you pressed “ENTER”, the file is now saved to the hard drive.
- 12) The maximum number of files allowed on the hard drive is 127.

MAKING A COPY OF A PROGRAM

- 1) Go to the “EDIT”, “FILE” screen
- 2) Press the soft-key under “copy” and move the cursor to the file you want to make a copy of and press “ENTER”.
- 3) It will prompt you in the lower left corner of the screen, input filename.
- 4) Type the name you want to call the copied version and it appears in the upper left corner and press “ENTER”.
- 5) It will prompt you in the lower left corner, will you copy?
- 6) Press “ENTER” for yes and “ESC” for no.
- 7) If you pressed “ENTER”, the same program now has 2 names.

LOADING PROGRAMS FROM USER DISK TO MEMORY (WITHOUT HARD DRIVE)

- 1) Insert user disk in drive 2.
- 2) Press the soft key next to ““EDIT””.
- 3) Press the soft key next to “FILE”.
- 4) Make sure the Load/Save button is highlighting the word “Load”.
- 6) Using the arrow keys, cursor to the file you wish to load and press “ENTER”.
- 7) The program is now in memory.

LOADING PROGRAMS FROM USER DISK TO MEMORY (WITH A HARD DRIVE)

- 1) Insert user disk in drive 1.
- 2) Press the soft key next to ““EDIT””.
- 3) Press the soft key next to “FILE”.
- 4) Move cursor up to filename on left side of screen by pressing the up arrow key (↑) on the keyboard.
- 5) Type C: and press “ENTER”.
- 6) The files appearing on the left side of the screen are the files on your disk in drive 1.
- 7) Make sure the Load/Save button is highlighting the word Load in blue.
- 8) Using the arrow keys, bring the cursor to the file you wish to load.
- 9) Press the “ENTER” button.
- 10) The program is now in memory.

LOADING PROGRAMS FROM HARD DRIVE TO MEMORY

- 1) Press the soft key next to ““EDIT””.
- 2) Press the soft key next to “FILE”.
- 3) Make sure the Load/Save button is highlighting the word Load in blue.
- 4) Using the arrow keys position the cursor to the file you wish to load.
- 5) Press “ENTER”.
- 6) The program is now in memory.

SAVING PROGRAMS FROM HARD DRIVE TO USER DISK

- 1) Insert user disk into drive 1.
- 2) Press the soft key next to “SET”.
- 3) Press the soft key next to “DISK”.
- 4) Press the soft key next to “USER COPY”.
- 5) Using the ↑ and ↓ arrow keys highlight the arrow pointing to floppy disk, and press “ENTER”.
- 6) If you want to save all the NC files on the hard drive to the user disk, press the (⇒) right arrow key to highlight all files to the right of NC program file and press “ENTER”.
- 7) Insert the user disk into drive 1 if you haven’t done so already and press “ENTER”.
- 8) All the NC files on the hard drive are now saved onto the disk.
- 9) If you want only specific files to be saved to disk, highlight specific files to the right of NC program file choice and press “ENTER”.
- 10) Files on hard drive will appear on the screen numbered in red, 16 maximum per

screen.

- 11) Choose the files you want to save to the user disk by typing the number in red which corresponds with the file you want and press the Return key.
- 12) There is a maximum number of 10 files you can save at a time.
- 13) To save the chosen files, press "ENTER".

SAVING PROGRAMS FROM USER DISK TO HARD DRIVE

- 1) Insert user disk into drive 1.
- 2) Press the soft key next to "SET".
- 3) Press the soft key next to "DISK".
- 4) Press the soft key next to "USER COPY".
- 5) Using the ↑ up and ↓ down arrow keys, highlight the arrow pointing to hard drive and press "ENTER".
- 6) If you want to save all the NC files on the user disk to the hard drive, press the ⇒ right arrow key to highlight all files to the right of NC program files and press "ENTER".
- 7) Insert the user disk into drive 1 if you haven't done so already and press "ENTER".
- 8) All the files on the user disk are now saved to the hard drive.
- 9) If you want only specific files to be saved to the hard drive, highlight specific files and press "ENTER".
- 10) Files on the user disk will appear on the screen numbered in red, 16 maximum per screen.
- 11) Choose the files you want to save to the hard drive by typing the number in red that corresponds with the file you want and press the Return key.
- 12) There is a maximum of 10 files you can save at a time.
- 13) To save the chosen files, press the "ENTER" button.
- 14) The maximum number of files on the hard drive is 127.

DELETING PROGRAMS FROM MEMORY (WITH OR WITHOUT HARD DRIVE)

- 1) Press the soft key next to "EDIT".
- 2) Press the soft key next to "FILE".
- 3) Press the soft key under "DELETE" to highlight it.
- 4) Using the arrow keys, move the cursor to the right side of the screen and highlight the program you want to delete from memory and press "ENTER".
- 5) The prompt, will you delete, appears in the lower left corner of the screen.
- 6) Press "ENTER" for yes or "ESC" for no.
- 7) If you want to delete all the programs in memory, highlight Memory Clear.
- 8) The prompt, will you clear memory?, appears in the lower left corner of the screen.

- 9) Press "ENTER" for yes or "ESC" for no.

DELETING PROGRAMS FROM USER DISK (WITHOUT HARD DRIVE)

- 1) Insert the user disk in drive 2.
- 2) Press the soft key next to "EDIT".
- 3) Press the soft key next to "FILE".
- 4) Press the soft key below "DELETE".
- 5) Using the arrow keys, move the cursor to the left side of the screen and highlight the program you want to delete from the user disk and press "ENTER".
- 6) The prompt, will you delete?, appears in the lower left corner of the screen.
- 7) Press "ENTER" for yes and "ESC" for no.

DELETING PROGRAMS FROM USER DISK (WITH HARD DRIVE)

- 1) Insert the user disk into drive 1.
- 2) Press the soft key next to "EDIT".
- 3) Press the soft key next to "FILE".
- 4) Move the cursor up to filename in the upper left corner of the screen by pressing the ↑ up arrow key on the keyboard.
- 5) Type C: and press "ENTER".
- 6) Press the soft key under "DELETE".
- 7) Using the arrow keys, move the cursor to the left side of the screen and highlight the program you wish to delete from the user disk and press "ENTER".
- 8) The prompt, will you delete?, appears in the lower left corner of the screen.
- 9) Press "ENTER" for yes or "ESC" for no.

DELETING PROGRAMS FROM HARD DRIVE

- 1) Press the soft key next to "EDIT".
- 2) Press the soft key next to "FILE".
- 3) Press the soft key below "DELETE".
- 4) Using the arrow keys, move the cursor to the left side of the screen and highlight the program you want to delete from the hard drive and press "ENTER".
- 5) The prompt, will you delete?, appears in the lower left corner of the screen.
- 6) Press "ENTER" for yes or "ESC" for no.

COPYING A USER DISK TO ANOTHER DISK (WITHOUT HARD DRIVE)

- 1) Press the soft key next to "SET".
- 2) Press the soft key next to "DISK".
- 3) Press the soft key below "USER COPY".
- 4) Insert the disk with your programs on it into drive 1 and the blank disk into drive 2.
- 5) Using the arrow keys, highlight the arrow pointing to drive 2 and press "ENTER".
- 6) If you want to copy all the NC files to drive 2, using the arrow keys, move the cursor to all files to the right of NC program file and press "ENTER".
- 7) All the NC files are now copied onto the disk in drive 2.
- 8) If only specific NC files are wanted, highlight specific files to the right of NC program file and press "ENTER".
- 9) Files that are on the disk in drive 1 will appear on the screen numbered in red, 16 maximum per screen.
- 10) Choose the files you want to copy by typing the number in red that corresponds with the file you want and press the Return key.
- 11) There is a maximum of 10 files you can copy at a time.
- 12) To copy the chosen files, press "ENTER".

COPY A USER DISK TO HARD DRIVE OR HARD DRIVE TO A USER DISK

- 1) Press the soft key next to "SET".
- 2) Press the soft key next to "DISK".
- 3) Press the soft key below "USER COPY".
- 4) Using the arrows keys, choose which way you are copying files and press "ENTER".
- 5) Insert your user disk into drive 1.
- 6) If you want to copy all the NC files, move the cursor to NC program file, all files and press "ENTER".
- 7) All the NC files would be copied.
- 8) If you want only some NC files copied, choose specific files and press "ENTER".
- 9) The NC files appear on the screen numbered in red, 16 maximum per screen.
- 10) Choose the files you want to copy by typing the number in red that corresponds with the file you want and press the Return key.
- 11) There is a maximum of 10 files you can copy at a time.
- 12) To copy the chosen files, press "ENTER".

USING THE EDITOR

Once you load a program into memory, you have complete control of the program. Many

editing functions can be performed. All editing functions require that the program be loaded into memory and then press the soft key next to "EDIT".

To search for information in memory:

- 1) Press the soft key below SCH / RPLC.
- 2) Move the cursor to select SEARCH UNDER THE WORD or SEARCH ABOVE THE WORD. This means search above or below where the cursor is currently located on the CRT screen. Press "ENTER" when ready.
- 3) Type the information you wish to search for.
- 4) Press "ENTER". If the info. cannot be found, it will prompt you in the lower left corner.
- 5) The cursor will move to the first location of the information.
- 6) To continue searching, press "ENTER". If no more info., it will prompt you.
- 7) To stop searching, press "ESC".

To replace information in memory:

- 1) Press the soft key below SCH / RPLC.
- 2) Move the cursor with the arrow keys to select REPLACEMENT OF THE WORD and press "ENTER".
- 3) Type the information to be replaced and press "ENTER".
- 4) Type the replacement information and press "ENTER".
- 5) Press (Y) for yes or (N) for next or "ESC" to stop..

To exchange information in memory:

- 1) Press the soft key under EXCHANGE/ INSERT so it highlights exchange.
- 2) Move the cursor to the information you want to replace with other information.
- 3) Type in the new data and the old data is replaced.

To insert information in memory

- 1) Move the cursor to the information AFTER where you want to insert data. (The inserted data will go before, where the cursor currently is).
- 2) Press the soft key under "INSERT" so it highlights INSERT .
- 3) Type the new data and it inserts the information.

To mass delete information in memory or to relocate that information elsewhere:

- 1) Search to the first character to be deleted.
- 2) Press the soft key under "AREASET".
- 3) Bring the cursor to the last character to be deleted (characters to be deleted will be highlighted).
- 4) Press the soft key under "DELETE".

5) Note: To return the deleted words, or insert them elsewhere in the program, or into another program in memory, move the cursor to the desired location and press the softkey under COPY. This key remembers only the last information area set highlighted.

To delete information in memory:

- 1) Move the cursor to the character you want to delete from the program.
- 2) Press the delete key located above the enter key on the keyboard and the character is deleted.

To delete file:

- 1) Press the soft-key under File Delete.
- 2) "Will you delete" appears in the lower left corner.
- 3) Press "ENTER" for yes or "ESC" for no.
- 4) This will delete the program you are viewing, from memory.

To include 2 programs together:

The program you want to include into another program must be in memory. Highlight "EDIT" and press the soft key below more in the lower right corner. Position the cursor in the proper location in the program and press the soft key below include and the files on the hard drive or floppy disk appear on the screen. Choose the file you want spliced into the file you have in memory by moving the cursor to it and press "ENTER". The file you chose is now spliced into the original program above where the cursor was located before you pressed include. You may need to now cleanup the program.

COPYING A SYSTEM DISK (WITHOUT HARD DRIVE)

Once you are up and running, you will want to be sure that you have spare copies of your important disk in case of accidental damage. We recommend having at least two backup copies of the SYSTEM disk (keep them in the office, away from the shop environment).

Note that any 3-1/2 inch disk double sided, high density, non formatted will work as the system disk. They can be found at almost any computer store.

Also, before doing this procedure it is wise to slide the write protect tab on the master system disk to the protected position. This will avoid disaster if you make a mistake during this procedure. When you are finished, you will want to store the master away in a secure location. Use the copy as your day to day working SYSTEM disk.

- 1) Press the soft key next to "SET".
- 2) Press the soft key next to "DISK".
- 3) Press the soft key under "SYSTEM BACKUP".
- 4) Place the BLANK, NON- FORMATTED disk in drive 2.

- 5) Place the master SYSTEM disk in drive 1.
- 6) Press "ENTER" .

Note: This procedure takes about 15 minutes.
Controls with a hard drive cannot perform this procedure.

INITIALIZING A NEW USER DISKETTE

The USER disk is where you can store programs. As time goes on, you will fill the USER disk with programs and need to use a new one. You should know that when initializing a new USER disk, the current version of the condition file is placed on the new USER disk.

NOTE: Pre-formatted IBM double sided, double density disks (720KB) do not need to be initialized with the procedure below.

Any useful programs like 0145 and 0165 etc. must also be saved onto the new user disk.

Note that any good quality, 3 1/2 inch double sided, double density disk (2DD) will work for the user disk. This type of disk will have the square window in the lower left corner only.

- 1) Press the soft key next to "SET".
- 2) Press the soft key next to "DISK".
- 3) Place the BLANK disk in drive 2 if you do not have a hard drive OR
Place the BLANK disk in drive 1 if you do have a hard drive.
- 4) Press the soft key under "USER FORMAT" and press the Y key.
- 5) Select dipro format (720 KB). Please note that the 1 MB standard format is NOT IBM compatible.
- 6) Press "ENTER" (initializing takes about 1 1/2 minutes).

Note: Be careful to use a NEW disk. If you use a disk that has data on it, this procedure will erase the data.

Note : Pre-formatted IBM double sided, double density disks (720kb) do not need to be initialized with this procedure. If you want to boot up the machine with the pre-formatted disk, be sure to copy the following files onto this disk. If you initialize a disk with the procedure above ,the files below are automatically saved onto the disk.

COND.DAT

OFFSET.DAT

STRING.DAT

RUNNING PROGRAMS FROM THE ACTIVE MEMORY

- 1) Load program into active memory.
- 2) Press the lower right vertical soft key so coordinate screen is showing.
- 3) Check conditional switches (dry run, single block, scale, mirrors, etc.).
- 4) Make sure the cursor is at the top of the program. If not, press arrow the arrow up key, then the home clear key.
- 5) Press "ENTER" to start the program .

RUNNING PROGRAMS FROM DISK (WITHOUT HARD DRIVE)

- 1) Press the lower right vertical soft key so the file names appear.
- 2) Using the arrow keys, bring the cursor to the program name you wish to run and press "ENTER" and the program appears on the screen.
- 3) Check conditional switches (dry run, single block, scale, mirrors, etc.).
- 4) Press "ENTER" to start the program.

DRAWING GRAPHICS

The CNC Program must be loaded into Memory before this procedure is done.

- 1) Press the soft key next to "EDIT".
- 2) Press the soft key next to "GRAPH".
- 3) Set the graphic parameters the way you want.
- 4) Press the soft key under CUTNPASS SCRN (graphics should be shown).
- 5) If the alarm , same data are set in the T-P and T-N appears ,go back to "EDIT",GRAPH and look at T- program and T- Next. They cannot be the same. Set correctly and retry step 4.
- 6) If you get the alarm, UV presently not zero, the U and /or V axis display is not zero. Correct this by typing "G00UV" then "G92UV" in the "MDI" screen.
- 7) Once you get the picture on the screen press the soft key under FILE / SCRN SAVE (this **must** be done to allows you to see the graphics while burning).

Note: If the program has a M00 or M01 at the end of the NC line, it may not appear in graphics. If on a line by themselves , they will appear. See page 15-7 for more info.

Example:

(may not appear in graphics)

N0123 G01X12.354 M00

(will appear in graphics)

N0123 G01X12.354

N0124 M00

CHANGING POWER SETTINGS WHILE MACHINING

In the RUN mode, you have complete control of all condition parameters (ON, OFF, IP,,HP,MA etc.) while running a part. However, when you start changing these parameters, remember that you will be also changing important attributes of what the condition will do (accuracy, finish, size, etc.).

- 1) "RUN" a program.
- 2) Press the soft key under the condition parameter you wish to change (parameter will highlight in blue).
- 3) Type the new parameter value.
- 4) Press the soft key under the condition parameter (again) to enter it.

Note: If condition change in the "SET" screen is 0 any changes made are not saved (the next time this C condition is called, the original setting returns). If condition set is on 1, then the condition file changes.

When condition change is on 1, the change is only made to the RAM. When the machine is rebooted, the condition file on disk is loaded to memory (original file). So if you want the change to be permanent, then save the condition file using the "EDIT", "FILE" screen.

SWITCHING BETWEEN MACHINE CONTROL AND "APT"

Switching from machine control to APT programming.

- 1) From the CNC side Main Menu, press the soft key next to UTY.
- 2) Press the soft key next to APT.
- 3) At the prompt, insert the APT disk in drive one (remove the system disk first).
Delete this step if you have a hard drive.
- 4) Press the ENT key.
- 5) Press the F1 key.
- 6) At the USER DISK prompt, press the RETURN key.
- 7) The control will show a list of programs. If this is correct USER DISK, press Y to say yes.

Switching from APT to the machine control:

- 1) From the initial page of the conversational side (with basic menu), type 10 for "End of APT Function".
- 2) Replace the SYSTEM DISK in drive one when prompted to and then press return.
press "ESC" if you have a hard drive.
- 3) Press the F10 key when prompted to return to the CNC side.

COPING FROM DRIVE 1 TO DRIVE 2 (WITH HARDDRIVE AND 2 FLOPPY DRIVES)

- 1) Insert the disk you are copying from into drive 1.
- 2) Insert the disk you are copying to into drive 2. (It must be a formatted disk)
- 3) Press the soft key next to UTY.
- 4) If you have a software version earlier than 3.01AAB go to step 6.
- 5) If the version of software is 3.01AAB or later follow steps 1,2,3 then continue with step 7.
- 6) Press the soft key next to APT and go to step 8.
- 7) Press the soft-key below “to O S” to highlight it and press return and go to step 8.
- 8) Type copy C: *.* D: and press return .
- 9) Files that have been copied appear
- 10) Press F10 to return to the system.

Note: is the space bar .

COPYING ALL “APT” FILES FROM THE FLOPPY TO THE HARDDRIVE

Many customers have updated their machines to have a hard drive, or bought a new machine with a hard drive. The “APT” part , macro and nc programs from prior jobs may be located on the “USER” floppy disks which were used when programming those jobs. To copy files ,follow these procedures .

- 1) Press the soft keys “UTY” and “APT”
- 2) Put the floppy disk with “APT” files into drive #1
- 3) The dark line in step #3 is the space bar on the keyboard.
- 4) Type: copy c:*.pro _b: and press return. All files will be copied to the hard drive.
- 5) Press f1 to go to APT or f10 to return to the machine system.

Notes: _ symbol means the space bar.

For all macro programs use *.apt, instead of *.pro in step 4 above. (c:*.apt _b)

For all G code programs use *.nc, instead of *.pro in step 4 above (c:*.nc _b)

For all part programs use *.pro, instead of *.pro in step 4 above (c:*.pro _b)

COPY ALL PART PROGRAM FILES FROM THE HARDDRIVE TO FLOPPY DRIVE # 1

- 1) Press the soft keys “UTY” and “APT”
- 2) Put the floppy disk for the “APT” files into drive #1
- 3) Type: copy b:*.pro _c: and press return. All files will be copied to the hard drive.
- 4) Press f1 to go to APT or f10 to return to the machine system.

CHAPTER 15 CRT SCREENS

SCREEN DEFAULT PARAMETERS

This chapter looks at some of the CRT screens in the control. The screens shown in this manual describe some of the most common parameters used. This manual is not meant to replace the **“CONTROL SYSTEM”** manual supplied with the machine. The latest version of the Control System manual is volume 3.0A (p.n. 6100259). This manual is available from the parts department.

On the “SET” screens there are parameters which are considered preference settings. These affect the way the machine acts when you graph, dryrun or cut a program. The operator should be aware of the most common ones and what they affect and then decide how he wants the machine to act **and set them accordingly.**

NOTE: After you change any of the parameters, the changes are saved automatically when you **exit** the screen you were working on. When the control is sourced on, it will boot up with the parameter settings set the way they were when the machine was sourced off.

MAL-MDI SCREEN

X Y Z	0.000	U	0.000	EDIT
	0.000	V	0.000	
	0.000	W	0.000	
FILE :		ST	Flush	DSPLY
	A00			SET
		SPEED :	0.000	UTY
		OFFSET :	+000000000	
		TAPER :	+000000000	
				CUT'G
				M·T
PARAMTR : C O O O INSERT				
DRY	SINGLE	SCALE	PARAMTR	START
0	OFF	00001000		G30
				ON
				ON
				START
				RESET
				MORE

The “MAL – MDI” screen is blank in the middle allowing the operator to manually type in commands and programs.

Choose more in the lower right corner with the key below it and then choose load and the files on HD or userdisk appear in a box. Move the cursor up or down to choose the file you want to load. Once you find it, press “ENT” and it appears on the screen. Press “ENT” to execute the program.

To type information into the open screen, the word “INSERT” must be highlighted at the bottom of the screen. Use the “INS” key on the keyboard to toggle between insert and exchange. If you want to save this information, choose more and save. Input filename prompt appears in the lower left corner, so you must name the program now, using a maximum of 8 characters. After typing it in, press “ENT” and the information will be saved to the hard-drive or to the user disk.

If the information you typed in does not need to be saved, choose delete. Will you delete prompt appears in the lower left corner. Press “ENT” for yes and “ESC” key for no.

If you typed in some information, executed it and then wanted to bring it back, choose “MORE” to go to the second screen. Press “BACK” and it turns blue. Press the “BACK” button again and it will bring back the last information you typed in and display it on the screen.

MAL SCREEN

Choose “MAL” and choose “MDI” to UN-highlight MDI mode. This screen displays the functions available in manual (MAL) mode in the middle. It is another way to do the things you usually do in the “MAL”, MDI” screen. For example: To get the machine to

move to a location in the Y-axis, move the cursor to positioning move and press "RETURN" and it turns blue. The cursor moves to the X-axis. Move the cursor to the Y-axis and press the minus key. Enter the Y-axis position value then press "RETURN" and it turns blue. Now, press "ENT" and the machine will move to the new value. You can move several axes at once.

The next function is "CHNG-CRD 00". Move the cursor to "CHNG-CRD", type the screen number you want to change to and press "RETURN" and it turns blue. Now press "ENT" to change to that run screen.

The next function is "SHIFT". Move the cursor to "SHIFT", press "RETURN" and it turns blue. Now move the cursor to the axis you want to change and type the new value, press "RETURN" and it turns blue. Press "ENT" to change the value of the axis. This is like G92 code.

The next function is "SENSOR TOUCH". Move the cursor to "SENSOR TOUCH", press "RETURN" and it turns blue. Move the cursor to the axis you want to edgefind in and choose + or - for direction of the edgefind and press "RETURN" and it turns blue. Now press "ENT" to edgefind.

The next function is "LIMIT". Move the cursor to "LIMIT" and press "RETURN" and it turns blue. Move the cursor to the axis you want to send to a limit switch and also choose the direction + or - and press "RETURN" and it turns blue. Now press "ENT" to send the machine to a limit switch.

The next function is "HALF". Move the cursor to "HALF", press "RETURN" and it turns blue. Move the cursor to the axis you want to half, and press "RETURN". Now press "ENT" and the machine will move to half the value that was in the axis you chose.

The next functions are "ABSOLUTE" and "INCREMENTAL". Move the cursor to either choice, press "RETURN" and enter to change the mode the machine is in

The lower menu at the bottom of the screen does several things. "FEED", "ST", "LIMIT" and "HALF" simply move the cursor to the function you choose. If you choose "ABSO", it will change between absolute and incremental. If you choose "CRDNTS", it will go to the next run screen. If you choose "A ALL", it will zero out all axis in all 59 run screens.

Press the softkey below "MORE" and another menu appears. This allows you to zero out an axis in the current run screen by choosing the one you want or all axis with the "ALL" key. Press "MORE" again and this menu will switch to the run screen that you choose. There are several other functions in the lower right corner. Move the cursor to "FILENAME:" and type the name of the file you want to run. Press "RETURN". Now press "ENT" to execute the program. The machine will search for the program in memory first. If it doesn't find the program in memory, it will search the hard drive.

Another function is "INT.CENTER". Move the cursor to "INT.CENTER" and press "RETURN". Now press "ENT" to execute the 0145 centerfind program.

Another function is "CORNER PICKUP". Move the cursor to "CORNER PICKUP" and press "RETURN". Press "ENT" to execute the "CORNER PICKUP" program. The 0165 program must be located on the hard drive to execute the program.

Another function is "PARAMETER". You can type in variables here when using the above functions. Example: Highlight Corner Pickup and press "RETURN" and the cursor moves to parameter. You can then type 4,.01,0,0 and press "RETURN" and press "ENT" to pickup corner 4 using .010" wire and the X and Y display will be zero when the center of the wire is on the corner of the part.

The last function we will discuss is "IGNORE TOUCH". Move the cursor there and press "RETURN" and it turns blue. Choose which axis you want to move, enter the value and press "RETURN". Move the cursor to positioning move and press "RETURN" and "ENTER" and the machine will move even if the wire touches something.

EDIT, GRAPH SCREEN

001KE03		M'AL																				
Taper data																						
T_UPPER	=+0000.0000																					
T_LOWER	=+0000.0000																					
T_PROG	=+0019.0000																					
T_NEXT	=+0000.0000																					
T_DRAW	=+0000.0000																					
Graphic data																						
Graphic format	=0 (3D-0,XY-1,XY,XZ-2)																					
View angle (X rotate)	=030 (0-360)																					
(Z rotate)	=030 (0-360)																					
M00 Drawing	=ON																					
Random Drawing	=OFF																					
No. of contour	=05 (0-99)																					
End line for each block	=ON																					
Limit check	=ON																					
NC program	=OFF																					
Manual scale	=OFF																					
X,Y MIN	=+0000.0000	+0000.0000																				
MAX	=+0000.0000	+0000.0000																				
<table border="1" style="display: inline-table; border-collapse: collapse;"> <tr> <td>FILE</td> <td>SCRN</td> <td>CUT</td> <td>N</td> <td>PASS</td> <td>FILE</td> <td>SCRN</td> </tr> <tr> <td>INPUT</td> <td>LOAD</td> <td>SCREEN</td> <td>SELECT</td> <td></td> <td></td> <td></td> </tr> </table>		FILE	SCRN	CUT	N	PASS	FILE	SCRN	INPUT	LOAD	SCREEN	SELECT				<table border="1" style="display: inline-table; border-collapse: collapse;"> <tr> <td>PARAM</td> <td>SCRN</td> <td>FLAG</td> </tr> <tr> <td>INPUT</td> <td></td> <td></td> </tr> </table>	PARAM	SCRN	FLAG	INPUT		
FILE	SCRN	CUT	N	PASS	FILE	SCRN																
INPUT	LOAD	SCREEN	SELECT																			
PARAM	SCRN	FLAG																				
INPUT																						
		ON/OFF																				

When you go to **EDIT, GRAPH, CUT N PASS** and you get the error message same data are set in Taper Data TP & TN, you will have to enter either **0** or **Workpiece Thickness** in table to Next under Taper Data when in Graphics. Now try to do a **CUT N PASS** again.

NOTE: Graphics drawing will show the **TOP** of the part as **BLUE**, and the **BOTTOM** of the part as **GREEN**.

GRAPHIC FORMAT: Choose **0** for 3D, **1** for Top view or **2** for a Front & Top view.

VIEW ANGLE: Usually both X rotate and Z rotate at 30°. This specifies the angle of rotation of Graphics in X & Z axis, when in 3D.

M00 DRAWING: Shows where your stop is located when **ON**.

RANDOM DRAWING: When in 3D, it shows only a single plane, usually **OFF**.

NO. OF CONTOURS: When in 3D, how many slices through thickness of part. This controls the resolution of the picture.

END LINE FOR EACH BLOCK: When on, the graphics drawing will show red lines at the end of each block of NC .

LIMIT CHECK: When **ON**, it will tell you if the machine will hit a limit switch, when running the program. This does not replace dry run. Remember to be at the start point before graphing the program.

NC PROGRAM: When **ON**, it will show the NC program in the upper left corner of the screen as it graphs the program.

MANUAL SCALE: When **OFF**, the control automatically scales the size of the picture to fit on the screen, usually **OFF**.

When **ON**, you must define the size of picture you want on the screen using the XY MIN, MAX parameters below this parameter.

EDIT, GRAPH, SCRN FLAG INPUT SCREEN

X-Y CHANGE <input checked="" type="checkbox"/> MIRROR X =OFF MIRROR Y =OFF MIRROR Z =OFF MIRR SWAP REVERSE 1 =OFF SKIP =OFF INIT INCH =OFF INIT G91 =OFF TAPER IGNORE =OFF NOT USE I0 =+000000000 NOT USE I1 =+000000000 CHECKING INTERFERENCE = 0 OPTIONAL STOP =OFF FIGURE ROTATION X =+000000000 FIGURE ROTATION Y =+000000000 COORDINATE ROTATION X =+000000000 COORDINATE ROTATION Y =+000000000 SCALE = 00001000 RAM LINK =OFF DIGIT = 1	A CIRCLE POINT =+0000.0020 STRING PATTERN = 0 BAD APT =OFF NEW CALCULATION =OFF OFFSET PRIORITY 1 =OFF OFFSET PRIORITY 2 =OFF IN CORNER OVER =+0000.0000 IN CORNER RETURN =+0000.0000 OUT CORNER OVER =+0000.0000 OUT CORNER RETURN =+0000.0000 APPROACH PATTERN = 0 L-L ANGLE = 01000 PRESS =OFF SEARCH_PATTERN =OFF AUTO T UPPER = 0 G74 PATTERN = 0 PRESS CONTROL AMOUNT =+0000.0000 G00 CORRECT CANCEL =OFF DISPLAY = 0 GRAPHIC SINGLE RUN =OFF	M AL <input type="button" value="M AL"/> <input type="button" value="DISPLY"/> <input type="button" value="SET"/> <input type="button" value="UTY"/> <input type="button" value="FILE"/> <input type="button" value="FILE"/>								
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 25%;">SCRNCOND</td> <td style="width: 25%;">FILESCRN</td> <td style="width: 25%;">CUTNPASS</td> <td style="width: 25%;">FILESCRN</td> </tr> <tr> <td>INPUT</td> <td>LOAD</td> <td>SCREEN</td> <td>SELECT</td> </tr> </table>			SCRNCOND	FILESCRN	CUTNPASS	FILESCRN	INPUT	LOAD	SCREEN	SELECT
SCRNCOND	FILESCRN	CUTNPASS	FILESCRN							
INPUT	LOAD	SCREEN	SELECT							

REMEMBER: This screen affects graphics, not the machine! The most commonly used parameters are listed below:

MIRRORS - Turn the mirrors on and press soft key under Cut N Pass to see the graphics.

CHECKING INTERFERENCE - Set at 0, 1, or 2.

While cutting, if there's a smaller radius than the offset amount on an inside corner, the wire will dig into your workpiece. This is usually undesirable.

AT 0 - The control ignores the situation & digs into the workpiece.

AT 1 - If that situation can happen, the control will stop graphing and tell you the offset will create interference to make you aware of the situation.

AT 2 - The control decides how to correct the situation and graphs the program accordingly without warning you. You must then decide if the correction is acceptable.

SCALE - Scale factor 1000 = full scale should be set the same as scale in SET screen.

DIGIT - Chooses between 4, 5, or 6 place decimal when running in inch mode. Should be the same as digit on the set, travel screen.

TAPER IGNORE - If taper is in your program and this is turned on, Graphics will not show taper.

A CIRCLE POINT - When using G2 & G3, it compares the radius at the start and end points. If within the value specified in this parameter, the program will graph.

APPROACH PATTERN - Affects how the graphics enters and exits the geometry at 90° intersections.

1 - Leads straight in and stops short the offset amount & leads out at 90°.

0 - Leads in at an angle, and leads out at an angle.

GRAPHIC SINGLE RUN - When turned on, it draws one line of NC code at a time in graphics.

Press the Return key to continue drawing.

DISPLAY SCREEN

ACTL POSI		Start		<div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">M'AL</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">EDIT</div> <div style="background-color: black; height: 15px; margin-bottom: 2px;"></div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">SET</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">UTY</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">ACHIEVED</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">MAINT</div> <div style="border: 1px solid black; padding: 2px; height: 20px; margin-bottom: 2px;"></div>				
X	0. 00000	X	0. 00000					
Y	0. 00000	Y	0. 00000					
Z	0. 00000	Z	0. 00000					
U	0. 00000	U	0. 00000					
V	0. 00000	V	0. 00000					
W	0. 00000	W	0. 00000					
Machine		CUTLENGT 0. 0000mm		<div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">ACHIEVED</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">MAINT</div> <div style="border: 1px solid black; padding: 2px; height: 20px; margin-bottom: 2px;"></div>				
X	0. 00000	SPEED	0. 0000mm/min					
Y	0. 00000	MACH TIME	000H00M00S					
Z	0. 00000	TIME	000H00M					
U	0. 00000	Version	EDM 4.00ccc					
V	0. 00000	SH	I/O : 0081 MT1 : 0080					
W	0. 00000	RLD	: 0000 IPG : 0000					
		SUB1	: 0000 SUB2 : 0F00					
		SUB3	: 0000 SUB4 : 0000					
PARAMTR : C000								
DRY	SINGLE	SCALE	PARAMTR	START	GRA. NC	GRA. DT	TIME	TIME
1	OFF	0000010000		G30	ON	ON	START	RESET

Press the soft key next to “DSPLY”. This screen will show machine movement relative to the program running under actual position in the upper left corner.

This screen will also show machine numbers in the lower left corner, which shows where the machine is at, from its home position. When a specific axis is home that axis will be zero under machine numbers. These numbers change only when an axis moves. If these numbers are written down when the machine is physically located at the start point or pickup point, you can rely on these numbers to make sure the machine is back at the proper location, if you are in doubt of something.

The screen will show in the upper right corner, machine numbers showing where the machine was located at when the “ENT” key was pressed to start a program. In most cases, the machine was located at the start point of the program so these numbers are called start.

The lower right corner shows the cutting length of the program. Rapid moves (G00) are not included in this value. The cutting speed will be displayed and also the software version in the control. The software can be upgraded when it is determined that a later version is needed. The (machine time) clock resets itself when the program is activated. The (time) clock can be reset by the operator by first stopping it by highlighting the time start function and then pressing the key again to stop the clock. Now press the time reset key. To restart the clock, press the key below time stop and then again to start the clock.

SET SCREEN #1

SKIP	= <input checked="" type="checkbox"/>	FIGURE ROTATION X	= +000000000	M'AL EDIT DSPLY <div style="background-color: black; width: 20px; height: 20px; margin: 5px auto;"></div> UTY CHECK DISK DATA
NO_MAN	= 0	FIGURE ROTATION Y	= +000000000	
BUZZER	= OFF	COORDINATE ROTATION X	= +000000000	
N_STOP	= OFF	COORDINATE ROTATION Y	= +000000000	
N_DATA	= 000000000	M03	= 0	
DRY	= 1	NOT USE 26	= +000000000	
SINGLE	= OFF	OPTIONAL STOP	= OFF	
X-Y CHANGE	= OFF	Z RETURN	= 01	
MIRROR X	= OFF	REVOLUTION IN	= 0	
MIRROR Y	= OFF	CONDITION CHANGE	= 1	
MIRROR Z	= OFF	OFFSET CHANGE	= ON	CHECK DISK DATA
DRY DWELL IGNORE	= OFF	AWT REFERENCE	= 0	
MDI_TOP_START	= OFF	TABLE TO LIMIT	= +0000.0000	
T85 INVERTER FREQ(MR25)	= 015	TABLE TO UPPER	= +0000.0000	
T84 INVERTER FREQ(MR25)	= 050	TABLE TO LOWER	= +0000.0000	
NOT USE 16	= +000000000	TABLE TO PROGRAM	= +0000.0000	
NOT USE 17	= +000000000	TABLE TO NEXT	= +0000.0000	
FIGURE ROTATION A	= +000000000	TABLE TO DRAW	= +0000.0000	
COORDINATE ROTATION A	= +000000000	NOT USE 39	= +000000000	
SCALE	= 00001000	TAPER IGNORE	= OFF	

Parameters scroll by pressing ROLL UP / ROLL DOWN.

MACHINED	CHARGE	TRAVEL	MOTOR	AXIS	STR/PTN COMP.	CUTTING PARAMETER	ON/OFF

REMEMBER: THIS SCREEN AFFECTS THE MACHINE, NOT GRAPHICS.

SKIP - This **manually** turns on or off the block skip slash code in the program. This is the G11- G12 code if you want to program it.

NOMAN - When at 1 and an M02 is read, the power is turned **OFF**; like pressing **E Stop** button. Dry Run must be **0** for this to work. At 0, the power stays on.

BUZZER - If **ON**, the buzzer sounds continuously until manually turned off by pressing **ACK** or go to set and turn it off.

N-STOP - When turned on, program runs until the line before "N DATA" number.

N-DATA - Insert sequence line of the NC program you want to stop on. Must have "N-STOP" turned on.

DRY -

- 0 = Dry run turned off.
- 1 = Dry run on without sensor touch.
- 2 = Dry run on with sensor touch.
- 3 = Dry run on with XY axis moves of "TABLE TO DRAW" Z plane. Use this to plot out the taper at different Z heights.

SINGLE - When **ON**, the machine executes one line of NC code and stops.

X-Y CHANGE - When **ON**, the X Axis becomes Y and the Y Axis becomes X:

MIRROR X - When **ON**, it flips the program in the X-axis.

MIRROR Y - When **ON**, it flips the program in the Y-axis.

MIRROR Z - Not Used

DRY DWELL IGNORE - If there are Dwell codes in your program and you don't want the machine dwelling when Dry Running the program, turn this on.

MDI TOP START - When **ON** and when in "**MAL**", "**MDI**" and "**ENTER**" is pressed, the cursor jumps to the top of the program in the "**MAL**", "**MDI**" screen before starting the program.

SCALE - Scale factor 1000 is full size, 500 is half size. "**FLAG3D1000**" code if you want to program it.

M03 - When turned **ON** and the wire AWT operation cannot thread at the present location, the machine moves to the next location.

OPTIONAL STOP - When there is an M01 in the program, the machine stops there when this is turned ON. "**FLAG2D0**"=OFF or "**FLAG2D1**"=ON code if you want to program it.

Z RETURN - For Power Z Axis **ONLY**.

At 0 - When the program is stopped by **HALT**, **M01** or **M00**, and the Z axis is moved and **RST** is pressed, Z returns to position.

At 1 - If M01 or M00 is read and the Z axis is moved and **RST** is pressed, the machine cuts with Z axis moved. If **HALT** is pressed and Z axis is moved and **RST** is pressed, Z returns to position before restarting.

At 2 - If M01 or M00 are read, and Z axis is moved and **RST** is pressed, Z returns to position before restarting. If **HALT** is pressed and the Z axis is moved and **RST** is pressed, it cuts with the Z axis moved.

At 3 - Z **NEVER** returns if moved from the original position.

CONDITION CHANGE

When a "**C**" code is in the program, and cutting conditions are adjusted during the cut (On, Off, MA, SV, SF, Etc.).

At 0 - No changes are saved anywhere. ("**C**" code is not updated)

At 1 - Changes are saved to that power setting while the program is running. ("**C**" code is updated). Changes are only saved to memory, not the disk or hard drive.

AWT REFERENCE

0 (G14)= Wire break recovery turned off (T90-T91 codes still active).

1 (G13)= Returns with C888 power.

2 (G15)= Returns without power, but uses feedrate set by C888.

TABLE TO LIMIT - See Chapter 8

TABLE TO UPPER - See Chapter 8

TABLE TO LOWER - See Chapter 8

TABLE TO PROGRAM - See Chapter 8

TABLE TO NEXT - See Chapter 8

TABLE TO DRAW - See Chapter 8

TAPER IGNORE - Turns **ON** or **OFF** taper capability when G51 or G52 are in your program.

If you want to cut straight, turn it **ON**. If you want to cut taper, turn it **OFF**.

SET, TRAVEL SCREEN #1

A CIRCLE POINT	= 00000000	PRESS	= OFF	<div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">M'AL</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">EDIT</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">DSPLY</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">UTY</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">CHECK</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">DISK</div> <div style="border: 1px solid black; padding: 2px; margin-bottom: 2px;">DATA</div>			
DIGIT	= 1	JOG PASS RETURN	= 0				
STRING PATTERN	= 0	SEARCH_PATTERN	= OFF				
INIT INCH	= OFF	FORMAT_STOP	= OFF				
INIT G91	= OFF	RAM LINK	= OFF				
LOGICAL/PHYSICAL	= ON	NOT USE CODE 0	= 0000				
MACHINING HISTORY	= 0	NOT USE CODE 1	= 0000				
RESERVE 8 (M)	= +00000000	NOT USE CODE 2	= 0000				
STROKE CHECK	= OFF	NOT USE CODE 3	= 0000				
TWO LEVEL RESTART	= OFF	NOT USE CODE 4	= 0000				
BAD APT	= OFF	AUTO T UPPER	= OFF				
NOT USE 12	= +00000000	TAPER DEG OR D.MS	= OFF				
NEW CALCULATION	= OFF	DWELL ANGLE	= 01000				
NOT USE 14	= +00000000	OFFSET PRIORITY 1	= OFF				
IN CORNER OVER	= +0000.0000	OFFSET PRIORITY 2	= OFF				
IN CORNER RETURN	= +0000.0000	OFFSET DATA SAVE	= OFF				
OUT CORNER OVER	= +0000.0000	TAPER DATA SAVE	= OFF				
OUT CORNER RETURN	= +0000.0000	PRESS CONTROL AMOUNT	= +0000.0000				
APPROACH PATTERN	= 0	MIRR SWAP REVERSE 1	= OFF				
L-L ANGLE	= 01000	MIRR SWAP REVERSE 2	= OFF				
Parameters scroll by pressing ROLL UP / ROLL DOWN.							
MACHINING CHARGE		MOTOR		AXIS	STR/PTR COMP.	CUTTING PARAMETER	ON/OFF

A CIRCLE POINT - When using G2 & G3, it compares the radius at the start and end points. If within the value specified in this parameter, the program will run.

DIGIT - Can be set at 0,1 or 2 which shows 4, 5, or 6 place decimal when running in inch mode. It should be set the same as digit in the Edit, Graph, *Scrn Flag Input Screen*. 5 and 6 place allow you to split a tenth on the offset amount.

STRING PATTERN – 1 activates the string file. Usually set to 1.

INIT INCH - Initialize Inch mode **ON** or **OFF**.

INIT G91 - Initialize Incremental mode **ON** or **OFF**.

APPROACH PATTERN - Affects how the machine enters and exits the geometry at 90° intersections. See page 7-3 for more details.

0 - Leads in at an angle, the offset amount and leads out at an angle.

1 - Leads in straight and stops short the offset amount & leads out at 90°.

JOG PASS RETURN -

0 - When the HALT key is pressed during operation of a program & the machine is jogged, and RST is pressed, the machine returns back in a straight line.

1 - It reverses the path that you jogged the machine to return back.

AUTO T UPPER - (*For Power Z- Axis*) When turned on, it automatically tracks the location of the upper head and stores that location in Table To Upper in the SET Screen.

TWO LEVEL OFF - Only available on certain versions of software. When turned on and the machine is executing a program, you can press off and it will allow you to edit

the program above or below the line it is executing. If you edit something above where the cursor is, it won't take effect. Press "RST" to restart the program.

TWO LEVEL RESTART - When on and the "HALT" key is pressed or a stop is in the program the machine stops. If Z is moved and "RST" is pressed, Z returns and the machine stops again; you must press "RST" again to continue cutting.

SET, TRAVEL SCREEN #2 (ROLL UP KEY)

G74 PATTERN	= 1	SOFTWARE LIMIT	(X+) =+1000.0000	MAL EDIT DPLY UTY CHECK DISK DATA
DRY AWT	= 0		(X-) =-1000.0000	
SINGLE PATTERN	=OFF		(Y+) =+1000.0000	
NOT USE 44	=+0000000000		(Y-) =-0000.0000	
G200 CORRECT CANCEL	=OFF		(Z+) =+1000.0000	
TYPE OF PRINT FORMAT	=OFF		(Z-) =-0000.0000	
NOT USE 47	=+0000000000		(U+) =+1000.0000	
CHECKING INTERFERENCE	= 0		(U-) =-0000.0000	
INTERPOLATION JUDGEMENT	=+0000.0035		(V+) =+1000.0000	
INITIAL COORDINATE	= 054		(V-) =-0000.0000	
METRIC/INCH CANCEL	= 1		(W+) =+1000.0000	
CRDNT ROTATION CANCEL	= 1		(W-) =-0000.0000	
EDGE CONTROL CANCEL	= 1	RESERVE 73 (P)	=+0000.0000	
COORDINATE CANCEL	= 1	RESERVE 74 (P)	=+0000.0000	
MIRROR CANCEL	= 1	RESERVE 75 (P)	=+0000.0000	
X-Y CHANGE CANCEL	= 1	RESERVE 76 (P)	=+0000.0000	
SKIP CANCEL	= 1	SOFTWARE LIMIT IN/OUT	=OFF	
ABS/INC CANCEL	= 1	N-STOP A.A.R IGNORE	=OFF	
SOFTWARE LIMIT CANCEL	= 1	RESERVE 79 (D)	=+000000000!	
RESERVE 60 (M)	=+0000000000	RESERVE 80 (P)	=+0000000000	

Parameters scroll by pressing ROLL UP / ROLL DOWN.

MACHINESCHARGE	MOTOR	AXIS	STR/PTR COMP.	CUTTING PARAMETER	ON/OFF

DRY AWT –

This parameter affects how the AWT functions during dry run.

0 = The AWT simulation does not occur.

1 = The AWT pipe lowers down and the water jet turns on at each T91 location.

Depending on the software version it may even thread the wire.

CHECKING INTERFERENCE –

While cutting, if there's a smaller radius than the offset amount on an inside corner, the wire will dig into your workpiece. This is usually undesirable.

At 0 - The control ignores the situation & digs into the workpiece.

At 1 - If that situation can happen, the control will stop and tell you the offset will create interference to make you aware of it. **Recommended setting**

At 2 - The control decides how to correct the situation and the program continues without warning you. You must decide if the correction is acceptable.

SOFTWARE LIMITS - Enter the values needed to define the area of concern using machine numbers. For the machine to activate software Limits, go to "MAL", "MDI" and type G22 and press "ENT". To turn software limits off, type G23 and press "ENT".

SOFTWARE LIMITS IN/OUT -

ON - Keeps you from entering a predetermined cube when outside of it.

OFF - Keeps you from exiting a predetermined cube when inside of it.

This is the RS232 parameter screen.

SET, SIR/PTR SCREEN #2 (ROLL UP KEY)

Year
Month
Date
Hour (military time)
Minute

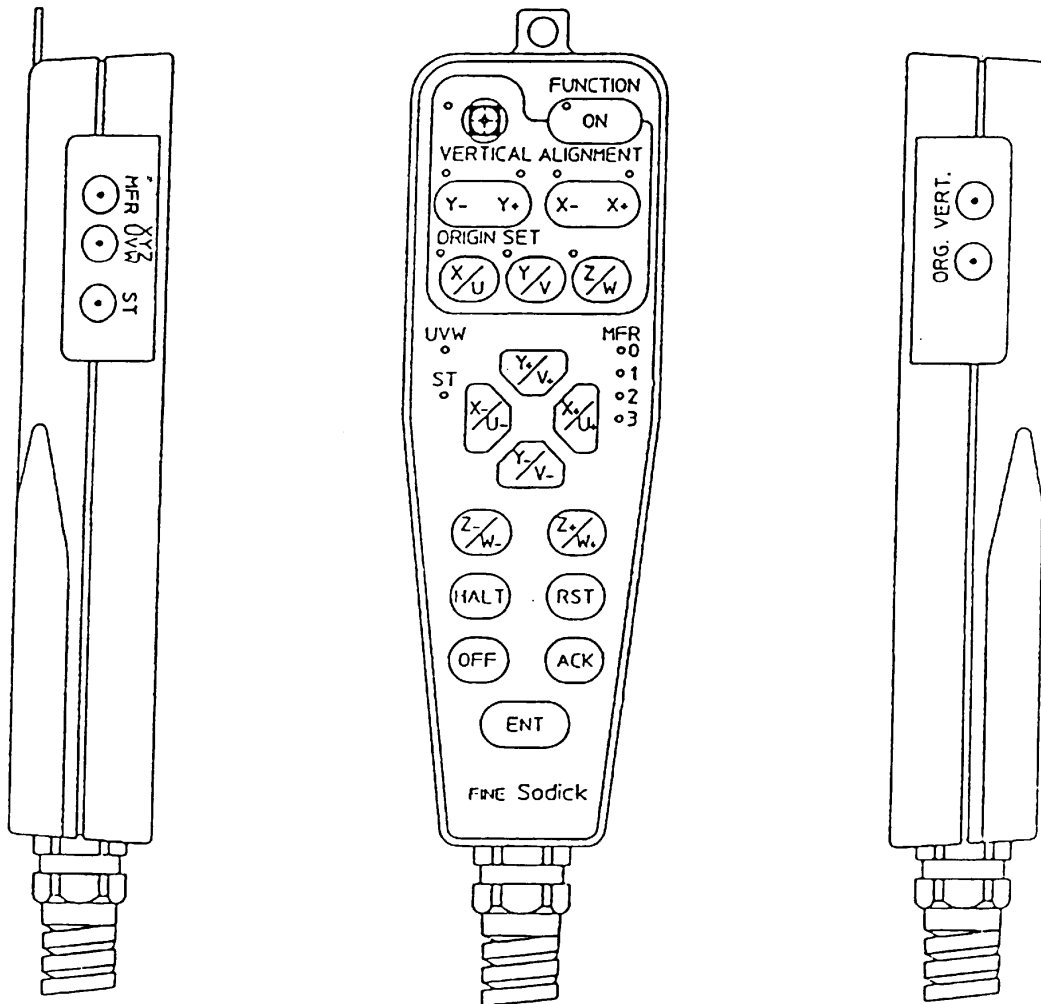
RUN SCREEN

X Y Z	<div style="font-size: 2em;">0.0000</div> <div style="font-size: 2em;">0.0000</div> <div style="font-size: 2em;">0.0000</div>	U V W	<div style="font-size: 0.8em;">0.0000</div> <div style="font-size: 0.8em;">0.0000</div> <div style="font-size: 0.8em;">0.0000</div>	<div style="font-size: 0.8em;">M'AL</div> <div style="font-size: 0.8em;">EDIT</div> <div style="font-size: 0.8em;">DSPLY</div> <div style="font-size: 0.8em;">SET</div> <div style="font-size: 0.8em;">UTY</div> <div style="font-size: 0.8em;">PORT</div> <div style="font-size: 0.8em;">GRAPH</div>																		
		<div style="border: 1px solid black; padding: 2px; display: inline-block;"> DRY: I </div> <div style="border: 1px solid black; padding: 2px; display: inline-block; margin-left: 10px;"> Flush </div>																				
		<div style="border: 1px solid black; padding: 2px; display: inline-block;"> SINGL </div>																				
<div style="font-size: 1.5em; font-weight: bold;">A00</div>		<div style="font-size: 0.8em;"> SPEED : 0.0000 OFFSET : +0000000000 TAPER : +0000000000 </div>																				
<div style="font-size: 0.8em;"> FILE : 001KE03 SINGLE </div>																						
C000 = 000 000 0000 000 0000 000 0 0000 0 000 000 000 000; C001 = 000 000 0000 000 0000 000 0 0000 0 000 000 000 000; H000 = +0000000000 H001 = +0000000000 H002 = +0000000000 ; H003 = +0000000000 H004 = +0000000000 H005 = +0000000000 ; G92XYUV: H000: M98P11111:																						
<div style="font-size: 0.8em;">PARAMTR : C100</div> <table border="1" style="width: 100%; border-collapse: collapse; font-size: 0.7em;"> <thead> <tr> <th>DRY</th> <th>SINGLE</th> <th>SCALE</th> <th>PARAMTR</th> <th>START</th> <th>GRA.NC</th> <th>GRA.DT</th> <th>TIME</th> <th>TIME</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>ON</td> <td>000010000</td> <td></td> <td>G30</td> <td>ON</td> <td>ON</td> <td>START</td> <td>RESET</td> </tr> </tbody> </table>					DRY	SINGLE	SCALE	PARAMTR	START	GRA.NC	GRA.DT	TIME	TIME	1	ON	000010000		G30	ON	ON	START	RESET
DRY	SINGLE	SCALE	PARAMTR	START	GRA.NC	GRA.DT	TIME	TIME														
1	ON	000010000		G30	ON	ON	START	RESET														

This screen is normally used to start and monitor the program. Press the bottom soft key, of the vertical keys. This key toggles between the run screen and a directory screen. The run screen displays the XYZUV and W axis readouts. The box below the UVW axis will display icons for various functions when they are not at the normal setting to remind you. The box below the icons will display the speed the machine is cutting, the offset being used and the taper angle being cut. Below the Z display is the file that is being executed. The capitol letter A in the middle of the screen means the machine is in absolute mode and the letter I means incremental. The numbers after the A or I represents what work coordinate screen or (run screen) the machine is running in. There are 59 different run screens. The program will be displayed in the middle of the screen for viewing only. Below the program will be the condition number that is being used. There is a menu along the bottom of the screen that contains several choices. To choose a item, press the softkey below the choice and it turns blue. Some of the choices allow numerical values to be entered to alter the setting. In that case, highlight the choice, type the new value and then press the softkey to enter it the new value.

If parameter is highlighted, it shows the parameters that make up the cutting condition that is being used. Press the more key to view the remaining parameters. These can be changed while the machine is cutting. NOTE: If WT or WS are at 0 ,the knobs on the control panel are affecting wire speed and tension. If a number is entered ,the knobs are overridden. WP and WC are not used. Press the more key again to return to the first parameters screen or the return fey to return to the main menu. If start G30 is highlighted, press the enter key prompt appears up above where the name of the program is displayed. When enter is pressed, the machine moves to the G92 location. GRA.NC and GRA.DT is for graphics. When the picture of the program is on the screen, you can have the NC program on or off the screen by using the GRA.NC key. There is also other data that can be turned on or off with the GRA.DT key.

CHAPTER 16 REMOTE CONTROL OPERATION



SELECTING A JOG SPEED

To choose the MFR speed on the remote, press the **MFR** button on the LEFT side and watch the red LED's on the front of the remote change.

- **0 - Fastest**
- **3 - Slowest.**

Note: The **KYB/REM** switch chooses between the remote or keyboard “MFR” settings. If the switch is at KYB the remote still works, but it moves at the “MFR” speed set on the keyboard.

To JOG X OR Y

To jog the X,Y or Z Axis, press the buttons located in the middle of the remote.

To JOG U OR V

To jog the U or V Axis, press the **XYZUVW** button on the **LEFT side**. The “UVW” light on the front of remote turns on. Now press the desired axis jog button located in the middle of the remote. Press the button on the **LEFT side** again to turn the U & V Axis off.

To EDGEFIND

To pick up the edge of a part, turn the wire run on manually, press the “**ST**” button on the **LEFT side** of the remote and **hold it down**. Now jog the machine so the wire bumps into the edge your picking up. Once the wire bumps the edge, release both buttons and the edge of the wire is flush with the edge of the part. Remember the center of the wire is not on the edge at this point. Also, remember to turn the wire run off. The “**ST**” light on the front of the remote tells you when the wire is touching something by lighting up.

To CENTERFIND

To center find a hole, press the “**ON**” button to turn on the light. Now press the button with the cross hairs inside the circle. The wire run turns on and the machine will pick up the edges of the hole X+, X-, Y+, Y-, X+, X-, and stop with the wire in the center of the hole.

To ZERO X Y OR Z DISPLAY

To zero out the X Y or Z Axis on the CRT, press the **ON** button located inside the white border at the top of the remote. On the **RIGHT side** press the **ORG** button and hold it down. Now under origin set press the axis you want to zero out on the CRT and you will see the bottom of the CRT flicker and the axis will now be zero.

To ZERO U OR V DISPLAY

To zero out the U or V Axis, press the **XYZUVW** button on the **LEFT side**. The UVW light turns on. Press the **ORG** button on the **RIGHT side** and hold it down. Now under origin set, press the axis you want to zero out on the CRT and you will see the bottom of the CRT flicker and the axis will now be zero.

WIRE ALIGNMENT

To align the wire set your alignment block on the work table.

- 1) Go to “MAL”, “MDI”.

- 2) Press the **“ON”** button to turn on the light and functions located inside the white border at the top of the remote.
- 3) On the **RIGHT** side press and hold the **“VERT”** button .
- 4) Under Vertical Alignment, press the direction you want the wire to travel to spark on the block (do not use jog keys).

The C777 Condition file will be loaded into the generator and the wire run will turn on. The generator will turn on and the screen will go into Incremental.

- 5) When the above happens, let go of buttons. The wire will start moving toward the block.
- 6) Press the **“XYZUVW”** button on the **LEFT** side to switch it to **“U V axis”** mode, the UVW light turns on.
- 7) When the wire starts sparking on the block, jog the U or V axis to get a consistent spark from top to bottom of the block.

Use the **MFR** button on the remote to achieve the desired speed for jogging the U and V Axis during this procedure.

- 8) After the U & V Axis are aligned go to **“MAL”**, **“MDI”** and type **G90** and press **ENTER** to go back to Absolute.
- 9) Type **G97UV** and press **ENTER** to zero out the U & V Axis in all 59 work coordinate screens.

CHAPTER 17 SEVERAL COMMON ALARMS & MESSAGES

E111 Same Data are set in taper data T-P and T-N. - T-Prog and T-Next in EDIT, GRAPH screen cannot be the same. Make T-Next zero or workpiece thickness and redraw.

T-Prog and T-Next in the SET screen cannot be the same. Make T-Next zero or workpiece thickness and retry.

E161 Touching limit switch on stroke end overtravel. - The machine is at the end of its travel. Press ACK and jog the machine off the limit switch.

E164 Machine overmoved, release from the heat switch. - There is an overload on the servo motors or the machine's power button was OFF while the jog keys were pressed. You will have to source OFF for 5 minutes and reboot to eliminate the alarm. Press the ACK key and press the power off button and the source off key, then wait 5 minutes and follow the power up instructions on page 14-1.

H161 Dielectric level or service tank has dropped from the specified level. - If flushing turbulence bounces the tank level floats or if the flushing is set too low on a skim cut, you can get this alarm.

H164 Machining has stopped due to a short circuit. - The wire is touching something while trying to cut the program or has now touched something while cutting the part. Correct the problem and press RST.

H171 Physical error of AWT. - The brass block that holds the AWT pipe did not contact the stop block when it moved down during the thread cycle. The AWT slide may need the brass powder removed.

H175 Contact Detect. - If the wire contacts something, the alarm appears.

H189 Over the ST Band. - If the surface you are picking up is contaminated, the alarm will appear. Clean the surface and retry.

H197 Minus Machine Coordinate. - Go to the DSPLY screen and look under machine. An axis is showing a negative number. The machine is too close to a limit switch.

PUT 0 AT FOURTH DECIMAL POINT OF U,V AXIS

The U or V axis is not truly zeroed out. Go to MAL, MDI and type G92 UV and press enter.

SYSTEM ERROR #12 - The machine is lost and needs to be rehomed or your pressing ENTER with the machine to close to a limit switch. Go to DSPLY screen and machine numbers will be negative.

UV AXIS PRESENTLY RUNNING IS NOT ZERO - The wire is not square. Move the U and V axis to zero to square the wire and retry the procedure you're attempting.