SERIES 1100

STUDENT WORKBOOK

P/N 70000298

[illegible]

[illegible]

TABLE OF CONTENTS

SECTION FOUR: SETUP AT THE MACHINE

Setting Absolute Zero
Using the DRO Mode
Cancelling Tool Length Offsets
Setting a New Z Home Position
Setting Tool Length Offsets: Drilling
Setting Tool Length Offsets: Milling
Setting Tool Diameter Offsets
Machining the Part
QUIZ

SECTION FIVE: CALCULATORS

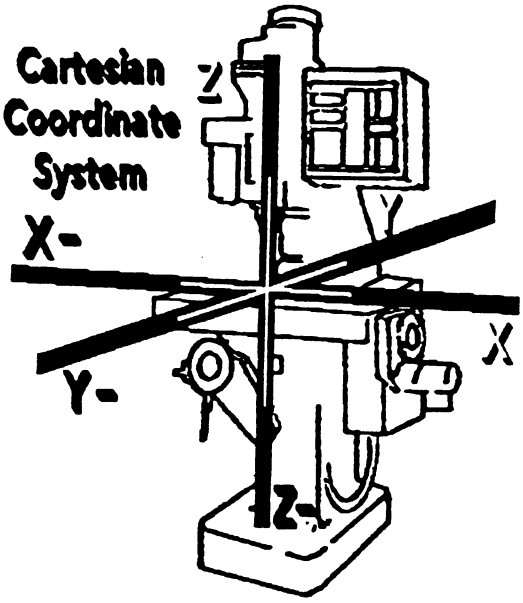
Math and Trig Calculator
Right Triangle Calculator
Geometry Calculator
Save and Recall
QUIZ

SECTION ONE:

FUNDAMENTALS OF PROGRAMMING

SECTION ONE: FUNDAMENTALS OF PROGRAMMING

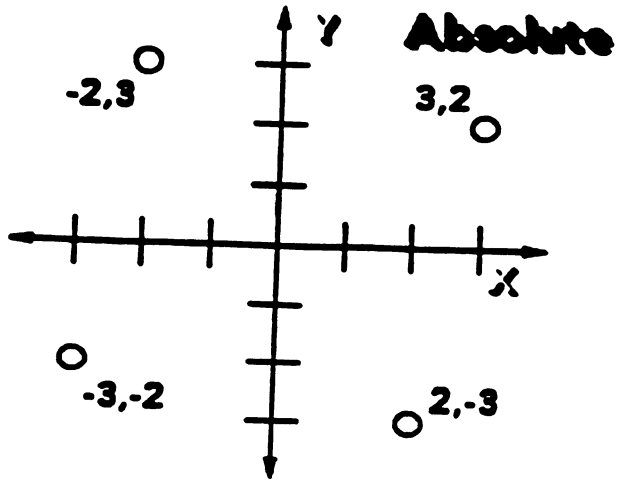
Welcome to Anilam Electronics' Series 1100 Training Program. In this training tape and workbook I'll take you step by step through the fundamentals of CNC programming and setup at the machine. Let's begin with a review of the basics of dimensioning theory.



AXES AND DIRECTION

A mill has three axes of motion, X, Y, and Z. The X axis runs back and forth, corresponding to the length of the part. The Y axis runs in and out, corresponding to the width of the part. And the Z axis runs up and down, corresponding to the depth of the part.

All dimensions are either positive or negative numbers. Any X axis dimension to the right of zero is a positive number. X axis dimensions to the left of zero are negative numbers. Any Y axis dimensions away from zero are positive numbers, and any Y axis dimensions in front of zero are negative numbers. And any Z axis dimensions above the workpiece zero are positive numbers, while Z axis dimension into the workpiece are negative numbers. The point where all three axes intersect is our zero reference point.



CARTESIAN COORDINATES

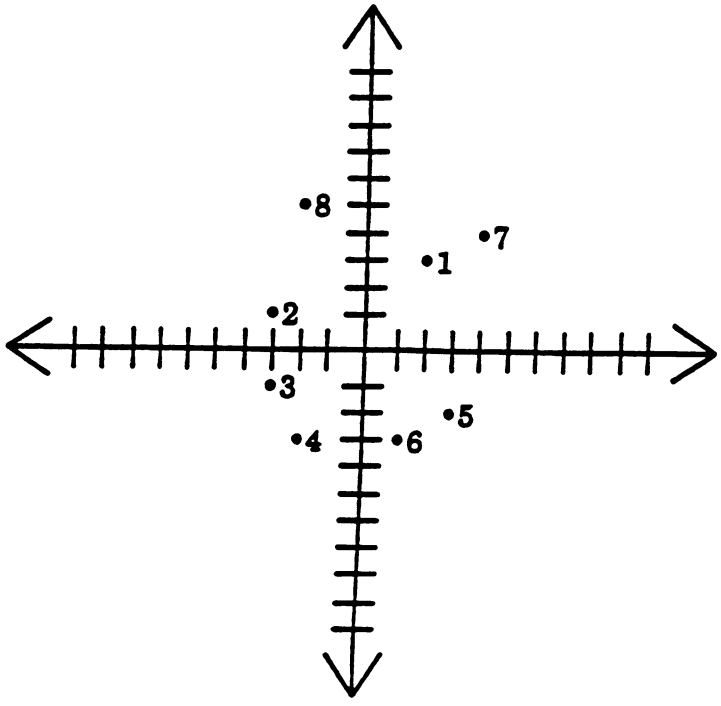
This system of measurement is known as the Cartesian Coordinate System, and it can be used to describe any point. For example, here is a center-referenced blueprint with four hole locations. Each hole location can be described as a specific X,Y position, taken from zero. Hole number one is at X positive 3, Y positive 2. Hole number two is at X negative 2, Y positive 3. Hole number 3 is at X negative 3, Y negative 2. And hole number 4 is at X positive 2, Y negative 3. These are called **absolute dimensions** because all are taken from an absolute zero location.

Another way to describe these locations is the distance they are from one another, or as **incremental dimensions**. Lets say the tool is at hole number one and we want to go to hole number two. The distance would be X negative 5, Y positive one, incremental. The distance from hole 2 to hole three would be X negative 1, Y negative 5, incremental. And the distance from hole number 3 to hole number 4 would be X positive 5, Y negative 1, incremental. And the distance from hole number 4 back to hole number 1 would be X positive 1, Y positive 5.

That's all there is to it. If you've done CNC programming before, you'll probably breeze right through this section and move directly to Section 3. But if this is new to you, make sure that you understand everything in this section thoroughly before moving on. Pay especially close attention to the part about positive and negative signs, and do all of the exercises in your workbook before moving on. If you need more help, rewind the tape and watch this section of the tape over again until you understand it.

QUIZ

SECTION 1:
FUNDAMENTALS OF PROGRAMMING



Referring to the above grid, circle the correct answers.

The X axis runs:	The Y axis runs:	The Z axis runs:
a. in and out	a. in and out	a. in and out
b. back and forth	b. back and forth	b. back and forth
c. up and down	c. up and down	c. up and down

Describe the X,Y coordinates of the following points:

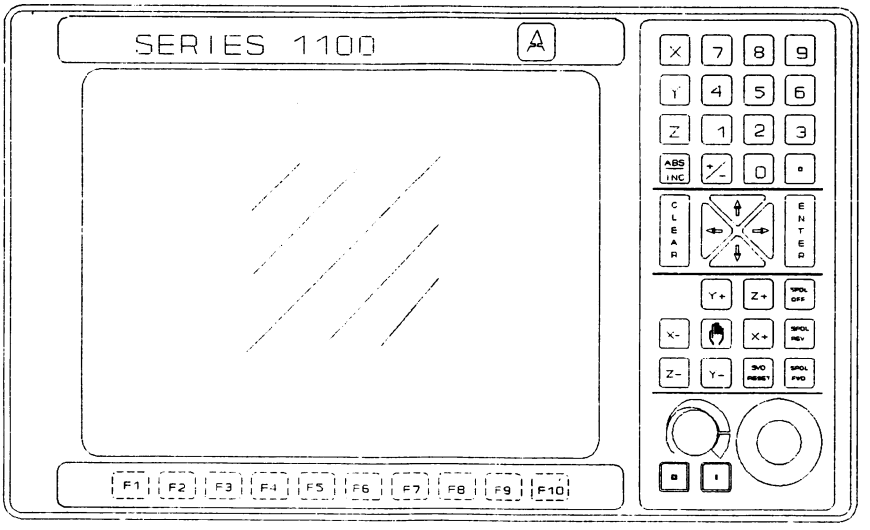
1.	2.	3.	4.
a. 3,2	a. 3,-1	a. 3,-1	a. 3,-2
b. 2,3	b. 1,-3	b. 1,-3	b. 2,-3
c. -3,2	c. -1,-3	c. -1,-3	c. -3,-2
d. -2,3	d. -3, 1	d. -3,-1	d. -2,-3
5.	**6.**	**7.**	**8.**
a. 2,-3	a. 1,-3	a. 4,4	a. -2,5
b. -3,2	b. -3,1	b. -4,4	b. 5,-2
c. 3,-2	c. -1,-3	c. 4,-4	c. -5,-2
d. -2,3	d. 3,-1	d. -4,-4	d. 2,-5

What is the incremental distance from point 5 to point 6?

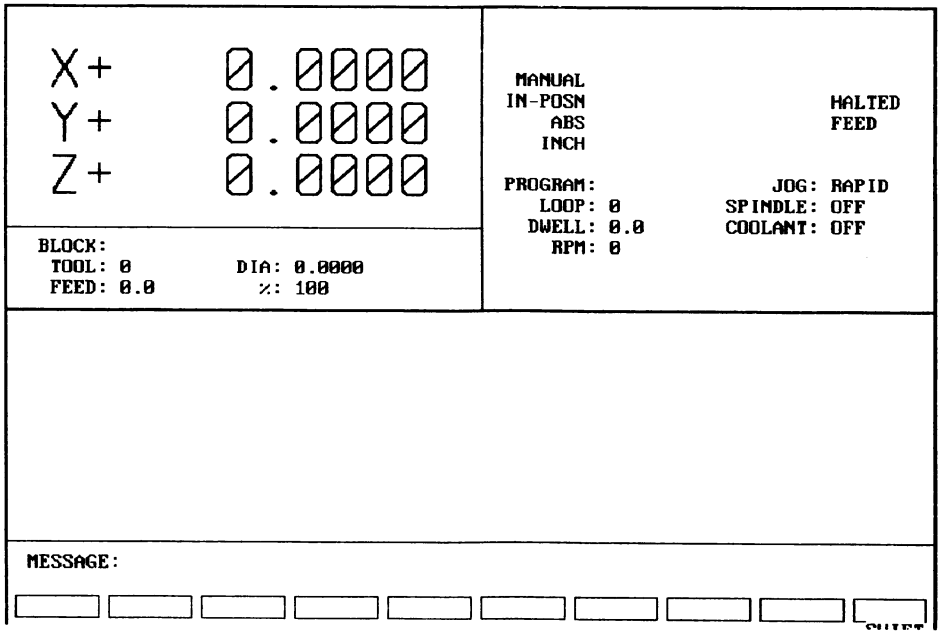
SECTION TWO:

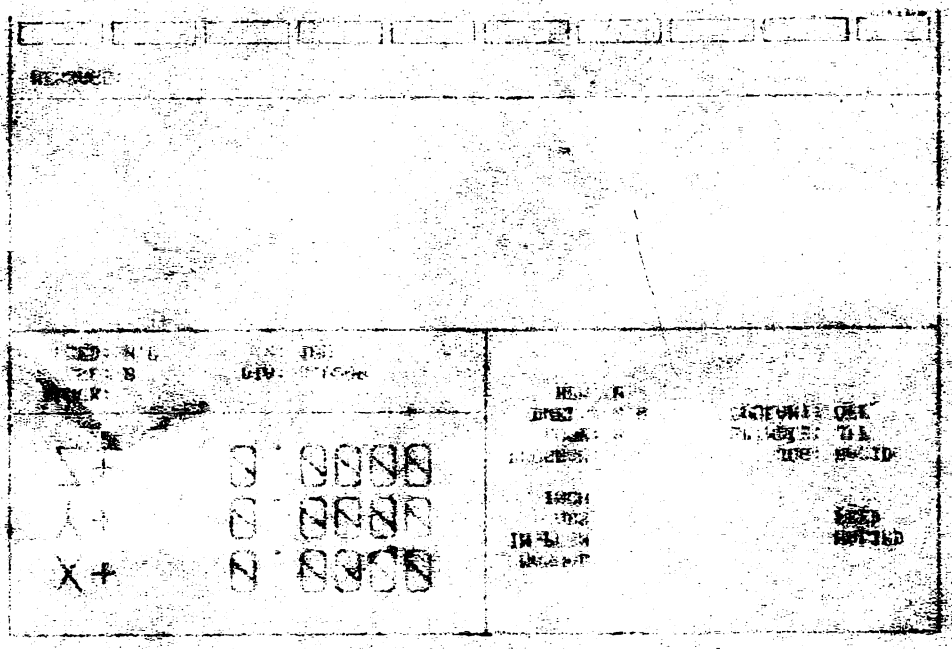
SERIES 1100 CONSOLE

S1100 CONSOLE

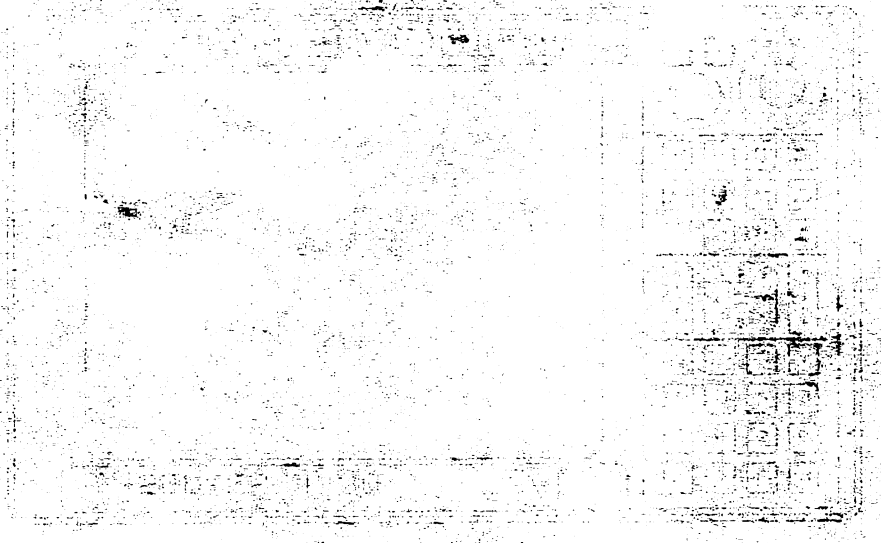


S1100 SCREEN





ST100 SCREEN



ST100 CONSOLE

CONSOLE OVERVIEW:

Now, let's take a closer look at the console itself. See fold-out.

Looking at the console, the faceplate is laid out with a keyboard and CRT screen. The keyboard is laid out into four sections: an alpha-numeric section, cursor movement section, a machine control section, and a section for starting and stopping the machine. The Series 1100 console also has a 14 inch monochrome CRT screen, which accesses and displays all of the control's programming functions and canned cycles. We'll start with an overview of the keyboard.

KEYBOARD LAYOUT

The alpha-numeric section at the top of the keyboard contains the XYZ keys and the numeric keys, here. We also have an absolute/incremental key and a +/- key.

The center section has a "clear" key, an "enter" key, and cursor movement arrows: left, right, up and down.

The section below it is used for jogging the machine in the manual mode. The "hand" key will select "rapid", "feed", "jog 100", "jog 10", and "jog 1". When you select the type of movement you simply select the X+ or X- key, Y+ or Y-, and in the case of a 3-axis Z+ or Z-. You also have a servo reset key for powering up the servo motors, and spindle keys if those functions are available on the machine.

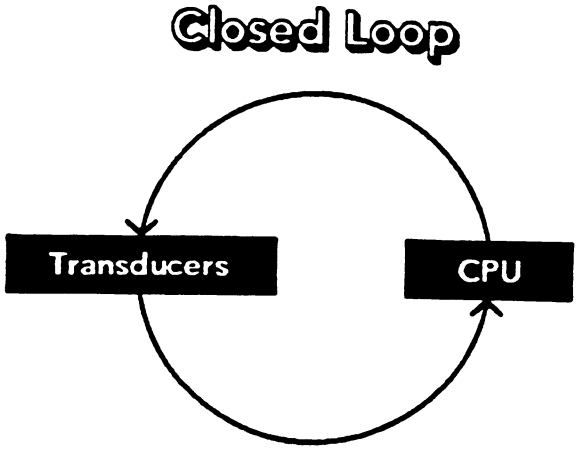
At the bottom of the keyboard, you have a section with a "feed rate override" knob, an "emergency stop" key, and "cycle start" and "cycle stop" keys.

CRT LAYOUT

The CRT screen is laid out so that you have the same screen whether you are in "manual", "single step" or "auto" mode. Here in the upper left corner is your axes position display. It tells you where the table is sitting in reference to absolute zero.

You also have two status areas. The small status area tells you what block number you are on, what the current tool number is, at what feed rate you're running, the tool diameter and the percentage of maximum feed from the feed rate override switch.

SECTION TWO: SERIES 1100 CONSOLE



The Series 1100 is a closed-loop system. In other words, it receives positioning information from highly accurate measurement transducers, compares the actual positions against it's programmed commands in internal memory, and simultaneously controls the speed and position of each controlled axis until each command is completed.

The Series 1100 is available in either 2 or 3-axis versions. It can cut arcs and angles... helical shapes... and 2 or 3 axis shapes of practically any configuration. It's fast. It's accurate. And in the hands of a skilled machinist it's capabilities are almost limitless.

On the other hand, it can't do your thinking for you. If you tell the control to rapid into the workpiece at 100 inches minute, it will try to do it. If you tell it to position to a point 6 feet past the end of the table, it will try to get there!

Of course, there are some built-in safety features to help cut down on major disasters. For example, if the servo motors try to push too far past their rated power limits, you will probably blow a fuse in the servo amplifier. And if you try to exceed the limits of the table travels of the machine, you will probably hit a limit switch and shut down power to the machine. But don't take any chances. Mistakes can be costly, and the best way to avoid making costly mistakes is to take your training seriously from the start and do it right the first time.

The upper right status area has quite a bit of information. It tells you which mode you are in, whether you are in absolute or incremental, inch or millimeter, whether you are halted or running, cutting an arc, a line or a rapid move. It also tells you what your current program name is. And if you are executing a loop, it will count down. Dwells will count down. There is an RPM display if it's installed on the machine. It also tells you whether you are in "rapid", "feed", "jog 100", "jog 10" or "jog 1". And it tells you whether your coolant is on or off.

The dark section below it will display the CNC program when you are in the "single step" or "auto" mode. To demonstrate, I'll power up the servos, go into "single step" mode, and you'll see the program appear, here. As the program is executing, the blocks will scroll up the screen.

Just below that section is the message line. Especially in two axis you will get messages to "position the Z axis and press START" or "prepare for a tool change and press START".

At the bottom of the CRT screen we have the soft keys. The soft keys are used most often in programming. They're used to select different functions. The box on the screen lines up above each soft key and that's how you select each one. Anything that is active at the time will be highlighted, such as the MANUAL mode soft key (F4).

SOFT KEY OVERVIEW

To get a better understanding of how the soft keys work, go into the EDIT mode and see what happens.

Looking at the soft keys, we now have a "teach in" mode which allows us to input manual positions into the program.

We have a "draw" mode which will allow us to display our program.

We have a "drill" window which will let us select "basic drill", "drill off", "pattern", and "bolt hole".

We have a "pocket" menu which allows us to program "rectangular pockets", "circular pockets", "frame milling", "hole milling", and "irregular pocket milling cycles".

We have a "mill" menu which allows us to select "rapid", "line", "arc", "z move", "set feed rate", "set zero", do a "home" function, do an "ellipse" or a "spiral". The "previous" key will bring us back to the previous menu.

We can call up a "tool number" or "cancel" a tool with soft key F6.

Our "calculator" keys are available here.

"Subprogram" brings up the subprogram mode where we can "define" a sub, "call" a sub, "end" a program or a subprogram, set up a "loop" to loop a subprogram, do "rotation", "mirror", or "scaling", set up a "timed dwell" or "dwell forever", and select an "M code" if M codes are installed on the machine.

The "misc." soft key allows us to "search" a block number (for example, "search" block 17). We can "page up" one page at a time, which consists of about nine lines, or "page down" one page at a time. We can quickly go to the "beginning" of the program or to the "end" of the program, which is especially handy in long programs. And we have a function called "quit". If you're in an editing session and you make a mistake and you just want to start over again without saving the program, you can select "quit". It will exit back out under the original program name without saving those changes.

QUIZ

SECTION 2:
SERIES 1100 CONSOLE

Excercise 1:

Look over the Series 1100 keyboard on your own. Locate the alpha-numeric section, the clear key, the cursor control keys and the enter key. Then locate the manual section and make sure you can identify each key and its function.

Excercise 2:

Carefully examine the CRT screen. Determine where the programming, positioning and tooling information are displayed. Then go into the manual mode and toggle through the five jog functions.

Excercise 3:

Locate the soft keys F1 through F10. Go through the layers of help for two or three canned cycles.

Questions: True or False?

Limit switches limit the maximum feed rates of each axis.

The Series 1100 is an "open loop" system.

A three axis Series 1100 can machine helical shapes.

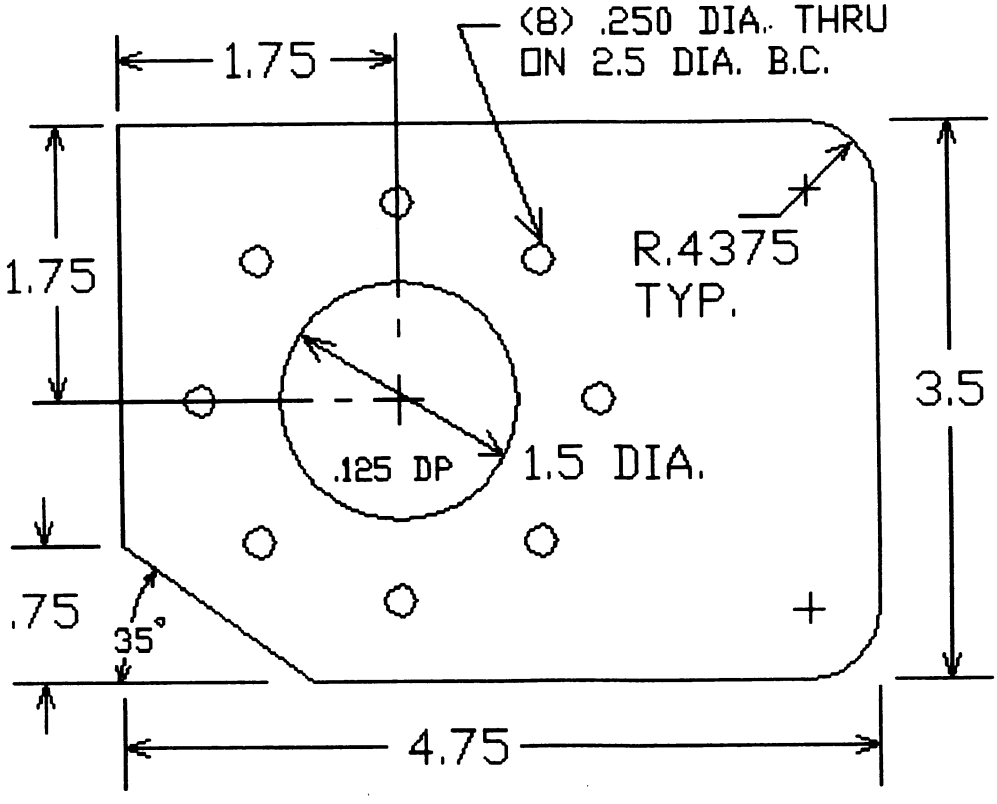
The bolt hole pattern canned cycle is one of the "drill" selections.

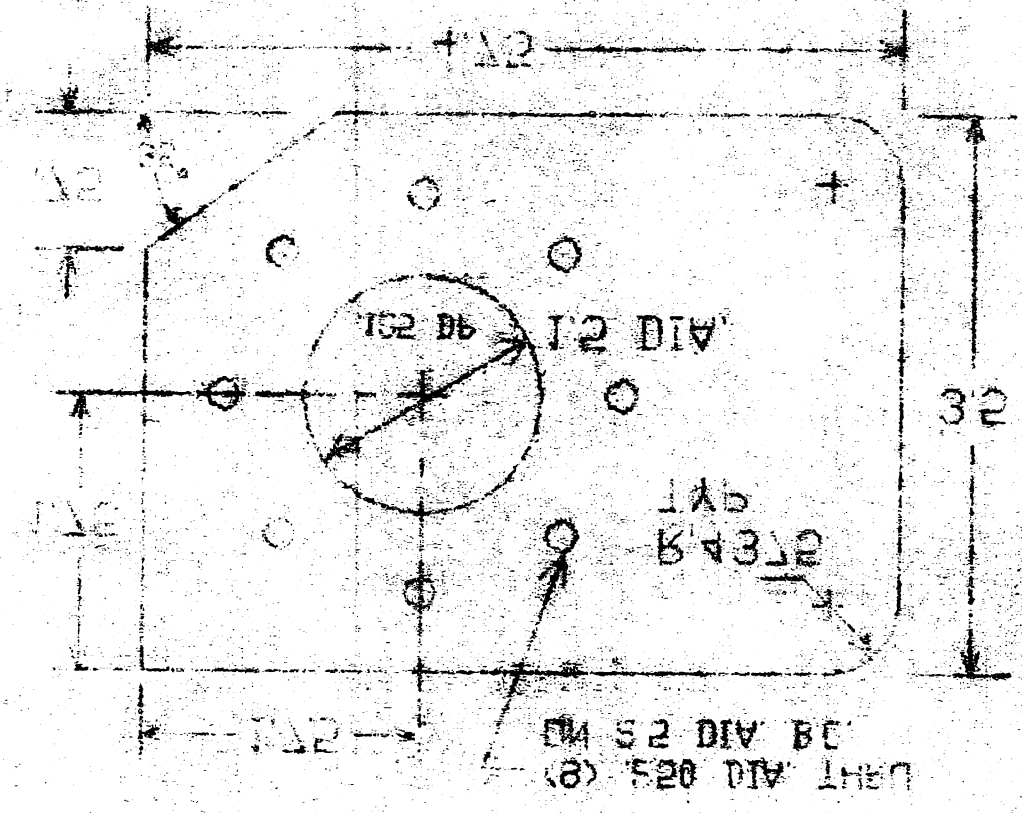
When pressed, the E-Stop button will halt all axes and power down the servo motors.

The soft key F6 is used to call up a tool number or cancel a tool.

SECTION THREE:

3-AXIS DEMO


(8) .250 DIA. THRU
ON 2.5 DIA. B.C.
1.75
1.75
R.4375
TYP.
3.5
.125 DP
1.5 DIA.
.75
35°
4.75



SECTION THREE: 3-AXIS DEMO

In this section, Dave McCarthy takes you through the programming process for the Series 1100 CNC control. This sample workpiece contains a contour around the outside, a circular pocket near the center of the part, and a bolt hole circle around that pocket. See fold-out.

SEVEN STEPS OF PROGRAMMING

In the logical programming of a workpiece, several things need to be decided before you even start programming, and other steps should be done before machining the part. Here are the seven steps of programming:

1. You decide how you're going to hold the workpiece, whether it be in a vise or a fixture.

2. Decide what tools you are going to use and in what order you are going to use them to machine the part.

3. Actually program the workpiece.

4. Draw the workpiece using "draw graphics" on the control to find any mistakes that you might have made.

5. Define your XY zero, or absolute zero, on the workpiece with an edge finder or an indicator.

6. Enter your tool length offsets, describing the distance from the bottom of each tool (in full retract position) to the top of the workpiece.

7. Machine the workpiece.

The first step is to decide how to hold the workpiece. In this example, a common table vise will be sufficient.

The second step is to decide what tools will be necessary for the job. There will be two tools required to machine this part. Tool number one will be a quarter inch drill. Tool number two will be a 7/16th, two flute end mill. The first thing that I want to do is drill the part. Then we'll come in and mill the part afterwards.

Now, we're ready for step three, programming the workpiece.

CREATING A PROGRAM NAME

First we need to make sure that we're in the MANUAL mode by using the soft keys at the bottom of the screen.

Next, press soft key F2, "program", where we can create, delete or edit programs. I'll press F2, "create a new program", and I'll give it a name. We can use the ASCII table to find the full alphabet, or I can just use the X,Y, Z keys and the numeric keys. I'll name this program "EX-2". So I search out the "e", enter it; search out the "x", enter it; and enter the "- 2" from the alphanumeric section of the keyboard. Turn off the ASCII table, and enter the program. The program is created and shown on the screen.

SETTING ABSOLUTE/INCREMENTAL DIMENSIONS

The next step is to go into the EDIT mode. Since its a new program, the screen will remain mostly blank, except for the soft keys at the bottom of the screen and the status area at the top of the screen.

The first step, as always when entering a new program, is to set your dimensions to "absolute". We do that with the "absolute/incremental" key. It can be toggled back and forth with the "+/-" key. We want to leave it at "absolute" so we press F10, save. F10 is always the "save" key, and we'll use it each time to enter each block into the program.

DRILLING CANNED CYCLES

Thinking about how we wanted to machine this part, we wanted to drill it first. So first we want to set up a drill cycle. In a three axis control we have to tell it how we want to drill. Whether we want to do a basic drill cycle, a peck drill cycle, or a boring cycle. I'm going to select the drill key, F3, and I'm going to select the "peck drilling" cycle. When I select it, the graphics and help menu comes up on the screen, and I fill in the blanks. Any entries that have zeros have to be filled in. The entries that are blank are optional entries.

Z depth:	-.3
starting heighth:	.100 above the workpiece
pecking distance:	.070

feed rate: 10 inches per minute
tool number: 1

Now that I've entered all the data, I press F10, save. The information is entered into the program, and the cursor advances down.

BOLT PATTERN CANNED CYCLE

Now that we've told the control how to drill, we have to tell it where to drill the holes. In the case of this blueprint, we have an eight hole bolt pattern, equally spaced around a two and a half inch diameter.

We have a drilling cycle especially for that. Its called a bolt hole pattern. Press F3, drill. Then select "bolt hole pattern". And again, we fill in the blanks and answer the questions.

X center: 1.75
Y center: -1.75
# of holes: 8
diameter: 2.5
starting angle: 0

We don't have to fill in the end angle, index angle or tool number. (We already activated the tool number in the first drill cycle.) So we press F10, save. The first hole will be drilled at zero degrees, and the rest of the holes will be equally spaced around the pattern.

Since there are no other holes to drill on the part, I have to turn the drill cycle off. We go into the drill menu and select "drilling off". And the process of drilling the holes is finished.

TOOL CHANGE POSITION

The next step will be to cancel the tool offset for tool number one, and raise Z to the home position. I'm going to press F5, mill. Select "rapid". Fill in

Z 0
Tool: 0

That's the Z home position. Save that.

The next logical step will be to move off to the side of the workpiece so that we can change tools. In this case I'm simply going to select a "modal" move. I do that by pressing the X, Y, or Z key.

I'll press X -1, Y 1.

Save that block. At that point the tool will go off the edge of the workpiece to make it easy to change.

CIRCULAR POCKET MILLING CANNED CYCLE

So we're ready to pick up the next tool now and begin our milling processes. We have a circular pocket to mill out, and we have a contour to mill around the outside of the workpiece.

We'll go ahead and mill the circular pocket first. In order to do that I'll back up one screen by hitting the "previous" key. And I see the "pocket" key. I'll select "pocketing", select "circular", and fill in the required entries from the help menu.

X center:	1.75
Y center:	-1.75
starting height:	.100 above the part
diameter:	1.5 inches
depth of pocket:	-.125
direction:	counter-clockwise (climb mill)
step over:	3
depth of cut:	(leave it blank, since we can take all the stock in one pass.)
finish stock:	.015
roughing feed:	15 inches per minute
finishing feed:	12 inches per minute
tool number:	2 (to activate tool number 2)

Press the F10, save key, and the block is input into the program. At this point the pocket will be completely milled and the tool will finish directly above the center of the pocket.

CUTTER COMPENSATION

The next step will be to make a rapid move off the upper left side of the part since I want to climb mill around the outside of the part. Climb milling around the outside of the part will entail using "cutter comp left". If I were to conventional mill around the outside of the part, it would be "cutter comp right". You always view the tool compensation, left or right, in the direction the cutter is moving. In this case, looking in the direction that the cutter is moving, I have to compensate to the left of the cutter path. So that's what I'll use.

I'll select the "mill" menu, F5. Then I'll select "rapid", F2. And I'll rapid off to the left of the workpiece, X -.5 and Y 0, which will be right on the Y zero line. The reason that I'm off on the X side -.5 is so that I can plunge the tool down in mid air without plunging into any material. I also want to activate "tool compensation left" at this point in time. Since we've already activated our tool number, we can leave the tool number entry blank. Save that block.

The next step will be to feed Z down to a cutting depth. So I'll select "line", and I'll enter Z -.06 with a feedrate of 20 inches per minute. Since I'm plunging in mid air, that's slow enough. Save that block.

CORNER ROUNDING

The next step will be to program our first line along the top edge of the part. Seeing that I have a corner radius on both the upper and lower right corners, I'm going to use the corner rounding function. No sense in me programming arcs there when the control has automatic corner rounding in it. So I'll select "line", and I'll fill in the full X dimension, which according to the print is 4.75. Then I'll put a corner radius on of .4375. I'll also slow the feedrate down to about 14 inches per minute. Save that block.

Next, we need to move the Y axis in a linear fashion. Select "line". The X doesn't change, so I don't need to fill it in. The Y we move to -3.5. Again, we put in a corner radius of .4375. Save that.

RIGHT TRIANGLE CALCULATOR

The next move will also be a line, but looking at the blueprint I don't know the X dimension. I know the Y dimension; it will remain at -3.5. But all that's known about the X dimension is that it's formed from a 35 degree angle with a .75 inch dimension given on the Y axis. I don't know the X axis point that I need to feed to right now.

I'm going to go ahead and select "line", and I'm going to use the calculator to find the X dimension to feed to. Press the "calculator" button, and select the center calculator which is a right triangle calculator. That's what we have here, a right triangle. So I'll select that, and the calculator comes on the screen. I have to imagine that the triangle is basically like the one shown on the screen. Side "a" of the triangle is .75 inches, so I key in .75 for the side "a" entry. I also know that angle "d" is 35 degrees in this, so I come down and fill in 35 for angle "d". Press "find", and the triangle is calculated and displayed on the screen. All of the sides and angles are filled in.

Side "b" is actually what I need. Side "b" happens to be 1.0711 inches. That's where I need to move my X axis to at this point. So what I'll do is place my cursor on side "b" and press "store", F9. Now, when I press "exit", that number is automatically filled into the X entry line. That's perfect; that's just what I wanted. So we'll go ahead and save that.

LINEAR INTERPOLATION

Now, we'll make another "line". This time the X has to move to zero, feeding along the angle. X is zero, and Y is -2.75 (3.5 minus .75). Save.

Now I want to make another line in the Y axis moving right off the workpiece. Select "line". Fill in a Y of a positive .5 inches, and save that block.

CANCELLING CUTTER COMP

After using tool compensation, you must turn it off after machining the last move of the contour. I just made my last line moving off the edge of the workpiece, so now I have to cancel tool compensation. That is done by creating one more line. Select "line", and move X away from the workpiece. I'll go back to where I started: X -.5

inches. And I'll toggle the tool comp to "off". Save it. Now I've fulfilled all of the needs of cutter comp: turning it on in a correct manner and turning it off in a correct manner.

The next logical step in the program will be to cancel the last tool and raise it up to home position. I'll select "rapid", Z zero, and tool zero. Save it.

MODAL MOVES

Then I'll make my modal move, which is simply an X or Y move with no rapid or line designation. It will be "rapid", since the last move that we inputted was "rapid". I can key in X -1, Y 1, and enter that. Since the last move was rapid, this move and all other moves will be rapid until I program another line or arc.

CHECKING THE PROGRAM

We're done with the program at this point. So the next step will be to use the cursor keys to take a quick review of the program, block by block.

In block one we set absolute dimensions.
We set a peck drilling cycle with tool one.
We drilled a bolt hole circle.
Turned off the drill.
Cancelled the tool, and rapided Z up to home.
Then moved off to a tool change position.
Next, we called a circular pocket with tool two.
Then we rapided off the edge of the workpiece with tool comp.
We brought Z down.
Made our lines around the outside of the workpiece.
And finally cancelled our tool compensation.
Cancelled our tool.
And moved off the workpiece.

DRAW GRAPHICS

It looks ok for now. The best way to tell for sure will be to go directly into "draw graphics". From the main edit menu, "draw

graphics" is F2. When I press that, the draw screen comes up, and I just press the "run" key. I'll turn the tool display "on" so we can see the tool drilling the bolt hole circle. Next, I see that tool two becomes active and it moves in to machine the circular pocket. I can also see the numbers change; this is called "simulation graphics".

I can toggle the tool on or off. If I turn the tool off, the program does draw faster. Any time that you drill holes with the tool active, you will get the cylinders along the screen to simulate the hole positions. I can also go to "single step" by hitting the motion key, and the tool will stop at the end of each block. To restart, I just hit the "start" key. I can also check my numbers at the end of each move. It looks good, and the program is finished.

Also note that the draw graphics is set to draw the part twice: once without tool compensation and once with tool compensation. The first drawing is the actual programmed tool path. The second drawing is the compensated tool path. If you put it into single step by motion, you can easily see by the numbers displayed on the screen that tool compensation is active. For example, at the end of the first move my X programmed dimension was 4.75. If you add half of the cutter diameter, the number comes out to 4.9687 inches, displayed for us on the screen.

It looked good in the isometric view. We can also take a quick look at it without text or tool in the XY view. I like to look at it in both those views. Its also possible to look at it from a front view or end view if you want to see the depth of the cuts. It looks alright, so we press "exit" to turn the draw mode off. "Exit" once again to back us out to the main program directory screen.

LOADING THE WORK PROGRAM

We need to press "load" to load our "ex-2" program as the work program. When we press "exit", we'll be all the way back out to the MANUAL mode. To machine the part, the next steps will be to find XY zero, or absolute zero, on the workpiece using an edge finder. Then we'll do our tool length offsets and cut the part.

QUIZ

SECTION 3: 3-AXIS DEMO

Questions: True or False?

After the program is written, the operator must decide how the workpiece will be held, what tools he will use, and in what order he will use them.

Absolute/Incremental dimensions can be toggled back and forth using the "+/-" key.

You have to tell the control how to drill before you tell it where to drill.

The F10 key is used to "save" a block.

When entering a canned cycle, all "blank" entries MUST be filled in.

You always view cutter compensation, left or right, in the direction that the tool is moving.

After using cutter compensation, one more move must be made to turn cutter compensation off.

To "draw graphics", press F2 from the edit menu, then press START.

Excercise 1:

Using the blueprint from this section, program the bolt hole pattern, circular pocket and contour on your own. Then use draw graphics to check the program. Use the step by step text in this workbook only if you get stuck.

SECTION FOUR:

SETUP AT THE MACHINE

SECTION FOUR: SETUP AT THE MACHINE

Now that we've finished programming the part and viewing it in draw graphics, we're going to go through the two set-up steps involved in setting up this workpiece at the machine.

SETTING ABSOLUTE ZERO

First of all, we have to set absolute zero, the X zero, Y zero point where all of our dimensions were referenced from on the blueprint. We decided that we were going to set it up in the upper left corner of the workpiece that we're holding in the vise.

So first of all, I'll take my .200" diameter edge-finder and get it up into the spindle. I'll set it at the proper heigth, and I'll just hand tighten it. Now I have to insure that my servos are on by resetting the emergency stop button and pressing "servo reset". That will turn on power to the servo motors.

Looking at the screen, I insure that my jog mode is in "rapid" by pressing the "hand" key. To jog the table into position, I simply press the X- button and the Y- button. Then I'll set my jog to "feed" (again, using the "hand" key) and press Z-. I'll move close to the workpiece by jogging in feed mode, overriding to a safe speed with the "feed rate override" knob down below, until I get near the workpiece.

I'll switch it now to "jog: 100", which is .010 at a time, and get to my depth and get near the edge of the workpiece. Now, I'll go ahead and turn my spindle on by hitting the "SPDL FWD" button. The spindle turns on in the forward direction. I jog carefully up to the edge of the workpiece in "jog: 100". When I get close, I'll switch it down to "jog: 10", which is .001 at a time. I'll move in .001 at a time until the edge finder kicks out of concentricity. I'll go ahead and turn off my spindle.

At this point, I know that the center of my spindle is .100 of an inch to the left of my workpiece. Therefore, my current position should read -.100. That's what I need to set in the X display. To do that I simply press X - .1 and press the ENTER key. The ENTER key is what sets zero or any other number in the display. So when I press ENTER, you'll see the X display change to X- 0.1000.

Now, I'll toggle my jog status back to "feed" and move away from the part in X. Then I'll come around to the solid jaw of the vise so I can set my Y dimension, raising Z up a little bit to clear the vise. I'll slow down the feedrate a little bit with the "feed rate override" switch, bring Z back down, and move in close to the Y positive edge of the workpiece. When I get close, I'll turn the spindle on and go to .010 jog at a time. When I get even closer, I'll switch to .001 at a time until the edgefinder kicks out. Then turn the spindle off. At this time, I know that I am sitting .100 to the positive side of Y zero. I'll touch Y .1 ENTER, and the display changes.

At this time I'll do a visual inspection. I'll raise Z up, switch my jog setting back to "rapid", press X zero, Y zero, and hit the START key. As opposed to the ENTER key, when I press START the machine will take off and move to my zero position in rapid fashion. So it rapided back to X zero, Y zero, and I can kind of do a visual inspection to make sure that the spindle is centered over the upper left corner of the part. In this way, I can make sure that I didn't make a mistake with my positive or negative signs when setting up part zero.

Now that we have our "absolute zero" set, all of our programmed dimensions will use this position as the zero reference. Now, I'll give it one more command. I'll press Z, zero, START, and the Z will also rapid back to it's zero location.

USING THE DRO MODE

Before we do tool length offsets, which is the next step, please note that setting your XY zero (and tool length offsets) can be done using the DRO mode. In other words, instead of using the servo power and pressing buttons, the hand cranks can be used. Make sure that the switch on the servo cabinet is set to the MANUAL position, and that the servo motors are off. By using the hand cranks, we can crank the table around and use the Series 1100 display just like a digital readout.

To set our tool length offsets, which is our next step, I'm going to turn on the servo motor drives and jog the machine using the control. First, I'll go ahead and take the edgefinder out of the spindle. Tool number one is a quarter inch drill. Tool number two is a 7/16ths end mill. In this case, these are not quick change holders, but to get

the point across, we're going to use them anyway. I'll get the collet up in the spindle, place the quarter inch drill in the collet, and tighten up the draw bar. This is tool number one in the program.

CANCELLING TOOL LENGTH OFFSETS

A very important note. Before the first tool length offset is entered, we must make sure that there are no previous tool length offsets active from a previous program. Say that you were running another program, and you stopped in the middle of it. A tool length offset could still be active here. What we need to do is look at the status block (under the readout display) and make sure that the tool block says "TOOL: 0". If it were to say tool one or tool 5 or any other tool, we must cancel that offset before we do any additional tool offsets. If it says "TOOL: 0", you know that there is no tool active.

Had there been a tool active, here, we could quickly get rid of it by writing a small MDI program. We could press the "MDI" soft key (F7), press the "tool number" soft key (F6), input a tool: 0, and EXIT out of the program. The message window says, "Press START to execute MDI of MANUAL to Cancel." So I press START, and if there was a tool length offset active, it will now be cancelled.

SETTING A NEW Z HOME POSITION

We have tool zero displayed in the status area. I'm setting at Z zero, and I'm comfortable with the height of the quill here for the Z home position. The Z home position (otherwise known as TOOL: 0, Z: 0) should be set near the top of the travel of the Z axis. It should be retracted all the way until it almost hits the limit switch. That's an ideal place to set Z home, the quill position with no tool active and Z at zero.

If the quill was not retracted far enough, we could easily set a new Z home position. Again, and most importantly, first of all we note that "tool zero" is active. Then we can set our new home position. I can jog the quill up, carefully, until I'm up near the top of the travel. In this case, I can see by the pointer on the quill when I'm nearly there. This new position will become our new Z home or "tool zero, Z zero".

Tool zero is active. So I simply press Z, 0, ENTER. Make sure that you press ENTER, not START. Pressing START would have driven it to Z zero. ENTER sets a new Z zero.

SETTING TOOL LENGTH OFFSETS: DRILLING

Our new Z home position is set, so now we're ready to set the tool length offset for tool number one. Using the "hand" key, I'll set my jog status to "rapid", and rapid to a position somewhere over the top of the workpiece. I'll override my feed, and I'll rapid down slowly near the center of the part. I'll switch to "jog: feed", and bring it down until I'm close to the top of the part.

One method of setting a tool length offset is to use a piece of paper, setting it between the tip of the tool and the top of the part. I'll then jog Z down .010 at a time, moving the paper back and forth until I get closer. Then I'll go to .001 at a time and jog down until the the tip of the tool pinches the paper so it can't be moved. The paper stops. Obviously, the tool is touching it. Since the paper is only a couple of thousandths thick, this method is plenty accurate enough for drilling a through hole.

Now, we're down at the top of the workpiece. The number shown in the Z axis display is the tool length offset for tool number one. The number is the actual distance from the tip of the tool at Z home to the top of the part when tool one is in the spindle. We want to enter this number into the tool length offset register for tool number one, so that every time that tool number one is active, the programmed Z zero reference location will be where the tip of the tool meets the top of the part.

The quick and easy way to get the tool length offset (displayed on the screen) into the tool page, is to press the "tool" soft key (F9). Cursor up to anywhere on the tool number one line. Then touch "calibrate Z". The tool length offset column for tool number one will be automatically filled in, and we can EXIT out of the tool page.

I'll make sure that my jog status is "rapid", bring the feed rate override back up, and I'll press Z, 0, START. Pressing Z, 0, START brings the tool back up to zero, the Z home position near the top of the quill travel. The tool length offset for tool number one is now complete.

SETTING TOOL LENGTH OFFSETS: MILLING

I'll remove tool number one from the spindle, jog away from the workpiece, and place tool number two in the spindle. Normally, in a CNC retrofit, you would have a quick change spindle so that the tooling will go back into the same position every time for every tool change. Therefore, the correct tool length offset is maintained every time after every tool change.

There is another method for determining tool length offsets, particularly for end mills where you need to be more accurate than with a drill. You can take some blue die, fan it out on the part and let it dry. Then move over the workpiece. So we jog over the workpiece. Switch to "feed", and bring Z down carefully. Go to jog 100 which is .010 at a time and bring it down even closer. Then turn on the spindle and jog down .001 until you see the blue die disappear. When it makes contact, turn the spindle off, and the tool length offset for tool number tool is displayed in the Z axis. Go into the tool page. The cursor is on the tool two line. So press "calibrate Z", and the tool offset for the second tool is finished.

SETTING TOOL DIAMETER OFFSETS

Before I exit the tool page I want to set my tool diameters. Tool number one has a diameter of .25. So I move the cursor to the "Diameter" column, punch up .250, and press ENTER.

Tool number two is a 7/16ths end mill, which is .4375 inches. Punch it in and press EXIT to leave the tool page. So our tooling information is now completed.

Finally, I'll raise the tool back up to our home position by making sure the jog status is in rapid, returning the feed rate override knob back to its normal position, and pressing Z, 0, START.

That completes our setup. We've set our XY absolute zero location in the corner of the part. We've set our Z home position, and we've entered our tool length offsets for tool number one and tool number two. The control holds up to 99 tool offsets, so there are plenty available for practically any application.

MACHINING THE PART

To machine the part, all we have to do is go into the AUTO mode, make sure that the desired CNC program is loaded into working memory, place tool number one in the spindle, and press START. The machine will automatically drill the bolt circle, wait for a tool change, then mill the pocket and contour when the START button is pressed again.

QUIZ

SECTION 4: SETUP AT THE MACHINE

Questions: True of False?

Before machining the workpiece, the operator must press "load" to load his selected program as the work program.

"Absolute zero" is always used as the tool change position.

The DRO mode is used to position the table using the hand cranks.

Tool zero, Z zero is also known as the Z home position.

Before the first tool length offset is entered, always make sure that the tool block status area says "TOOL: 0".

The "clear" key is used to cancel any previous tool length offset.

Once the proper Z home position is set, the quill can be fully retracted by pressing Z, zero, ENTER.

To enter a new tool length offset, position the tool at the top of the part, make sure the cursor is on the line containing the correct tool number in the tool page, and press "calibrate Z".

The Series 1100 can store up to 16 different tool offsets.

Excercise 1:

Use an edge finder to set absolute zero for the X and Y axes.

Excercise 2: (3-axis applications only)

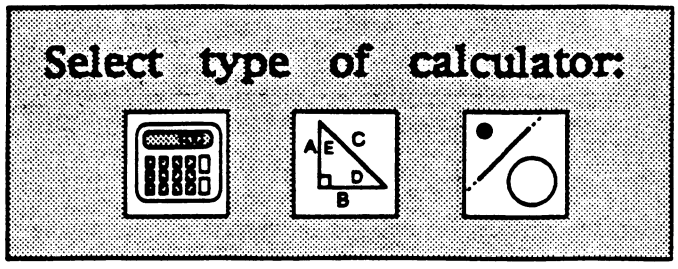
Set a new Z home position, then set the tool length offsets for tool number one and tool number two.

OPTIONAL: Machine the workpiece programmed in Section 3.

SECTION FIVE:

CALCULATORS

SECTION FIVE: CALCULATORS



The Series 1100 has three calculators which, when used properly, can save hours of programming time at the machine. All three calculators can be accessed from the "edit" mode by simply pressing the F7 key.

The first calculator is a "math and trig calculator" for adding, subtracting, multiplying, dividing, performing trig functions, etc. The second calculator is a "right triangle calculator" for figuring out sides and angles of triangles. The third one is a "geometry calculator" for inputting lines, points and circles, and for finding tough-to-define tangency points and intersections, much like a built-in CAD/CAM system.

All three calculators have "store" and "recall" functions, so that any XY coordinate found can be inserted directly into the CNC program that is being worked on. Not only is programming time and effort saved, but many potential keystroke errors are completely eliminated.

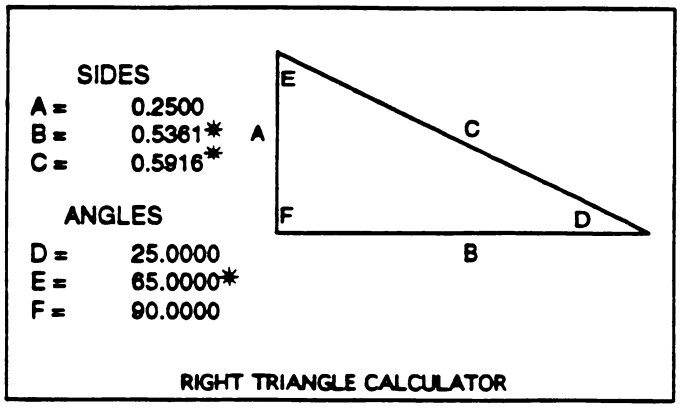
MATH AND TRIG CALCULATOR

We'll start with the "math and trig calculator". When we select it, it comes up on the screen and the soft keys change to: plus, minus, multiply, divide, left parenthese, right parenthese, function (including: sine, cosine, tangent, etc.), clear key, store key, and an exit key.

For example, we could key in "3.125 divided by 6" to find the increments between 7 holes, and when we press "enter" we get the answer on the bottom. We can then "store" that answer and insert it anywhere in the program, now or later on.

Clear that, and we can find the sine or cosine of a certain angle. For example, enter "35" degrees, "function", and select "cosine". The cosine of 35 is displayed at the bottom. Once again, I can "store" that answer for later use.

RIGHT TRIANGLE CALCULATOR



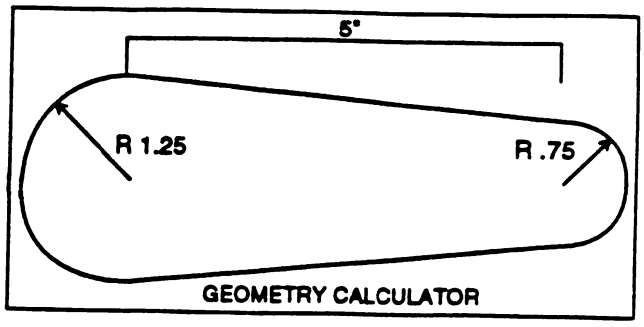
The second calculator is the "right triangle calculator". When we select it, it comes up with a window in the center of the screen, and the default triangle is shown in the upper left-hand portion of the screen. It labels side A, side B, side C, angle D, and angle E. Simply key in two pieces of information, and the calculator will do the rest.

Let's say that side A was a quarter inch long, and we knew that angle D was 25 degrees. We key in our two pieces of information, and press "find". The resulting triangle is drawn on the screen to actual scale, and all of the missing values are filled in for you by the control. So now we know that side B is .5361 inches long, side C is .5916 inches long, and angle E is 65 degrees. We can pick and choose any of those values to "store" in a separate little piece of memory to be inserted into the program later on.

GEOMETRY CALCULATOR

The last calculator is our "geometry calculator". The geometry calculator is used for inputting points, lines and circles, similar to a CAD/CAM system. The left-hand column of icons is used for inputting points onto the screen. The center column is used for

inputting lines. The column on the right is for inputting circles, with the exception of the last two definitions, which are used to input lines tangent to circles.



I'll give you an example of an oblong slot. We know the center point and radius of the first circle, so we can choose the "center and radius" icon. It prompts us for a radius value, so we enter 1.25. We then define the center of the circle: X 0, Y 0. The circle appears on the screen.

I'll repeat the process for the second circle. I select the "center and radius" icon again, and I'll enter a radius of .75 this time. The X center is 5 inches, and the Y center is 0. There's the second circle.

Now let's say that we want to put lines tangent to these two circles. I'll use the "line tangent to two circles" icon. Select it, and it asks the number of the circles to draw the line tangent to. The first circle is 1, and the second circle is 2. It shows us two tangent possibilities, and it gives us the option of choosing either one. Each one is numbered, so I'll choose number one.

Now, I want to put a line tangent on the bottom, I'll choose the same icon we used before. This time the line is tangent from circle 2 to circle 1. I select option 1 and the correct line is drawn for me.

Now, the ultimate goal is to find out exactly where the lines intersect the circles. So we go to the "point" definition showing the intersection of two elements. When I select it, it asks me for the element numbers. The circle is element one, and the line is element three. I select those elements, and the point is input automatically.

I repeat the process for the remaining tangency points until all of the intersections are found. Conventional CNC programming usually requires many point definitions, whether it's the center of a circle, the intersection of two lines, the intersection of a line and a circle,

etc. Therefore, the geometry calculator can save a lot of programming time, while eliminating the need for expensive offline programming systems.

SAVE AND RECALL

The "recall" function allows us to insert XY coordinates from the calculator directly into our CNC program. For example, when programming the first arc in the programming example shown above, simply select "recall", then select the geometry calculator. All the geometry that I saved comes up on the screen. If I select point number 5 and select "X and Y", both it's X and Y coordinates will be automatically input into the arc statement.

If I wanted to do a line to the next intersection, I would repeat the process. This time I would select LINE, "recall" from the geometry calculator, select point number 6, and choose "X and Y". The XY coordinates of point 6 are input directly into the CNC program.

I can continue to "recall" from the geometry calculator at any time, and input all necessary XY coordinates until my part program is completed.

Values can be recalled from any one of the three calculators. If I were doing a drilling operation and I wanted to recall from the first calculator, I could select one of the stored values, and input it onto the screen.

The same is true of the "right triangle calculator". If I were getting ready to machine an angular move, I could "recall" any saved value. In this case, I could select either angle E or side C from the stored values, and input them directly into the control. I could then SAVE the block in the program, or execute the block as an MDI move.

Not only can the calculators be used to find mathematical values or perform simple "CAD/CAM" functions, but the results can be inserted directly into the program without having to memorize them or write them down.

QUIZ

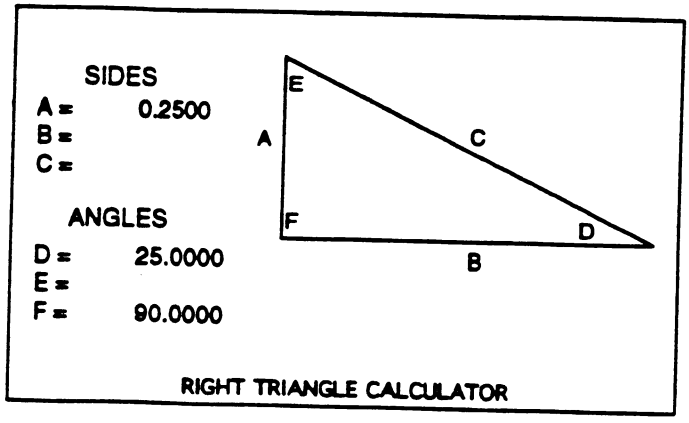
SECTION 5: CALCULATORS

Excercise 1:

Use the "math and trig" calculator to find the cosine of 35 degrees.

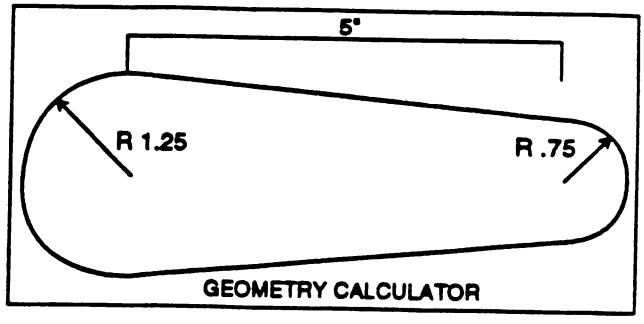
Excercise 2:

Use the "right triangle" calculator to find the missing sides and angles of the following triangle.



Excercise 3:

Use the "geometry calculator" to enter all of the points, lines and circles necessary to program the following oblong slot. Store all of the tangency points, then write a new CNC program using the stored information. After the program is written, use "draw graphics" to check your work.



CONCLUSION:

This completes the start-up training for the Anilam Series 1100 Control. By now, you should feel comfortable with the basics of dimensioning theory, 3-axis programming fundamentals, and setup at the machine. If you are having difficulty with any of these fundamentals, watch that section of the training video again and do all of your workbook examples. With a little practice, you should quickly master the basics.

It is also strongly recommended that you carefully review your **Series 1100 Reference Manual** at this time. The following sections will provide further instruction on such important applications as:

Section 6.6

- drilling
- milling
- pocketing
- subroutines
- loops
- mirror image, rotation and scaling

Section 7.4

- calculator examples

Section 6.6.5

- irregular pocket milling

Remember, the more that you learn about the system, the easier it will become, and the more productive you will be!