LEVIL 10M Control

0.1 LIST OF available CODES:

-G00: Rapid positioning.
-G01: Linear interpolation.
-G02: Circular interpolation clockwise.
-G03: Circular interpolation Counterclockwise.
-G04: Timer.
-G10: Skip function
-G17: XY plane.
-G18: XZ plane.
-G19: YZ plane.
G28: Go to Home position.
G43: activates + H offset.
G44: activates – H offset.
G49: cancels H offset.
G53: Machine coordinates.
G54: Work coordinates.
G55: Work coordinates.
G56: Work coordinates.
G57: Work coordinates.
G58: Work coordinates.
G61: Exact stop.
G64: Corner Round.
G80: Can cycle cancel.
G81: Drilling cycle.
G82: Drilling cycle with delay at bottom.
G83: Peck drilling cycle.
G84: Tap cycle (no rigid tapping).
G85: Boring cycle.
G90: Absolute positioning.
G91: Incremental positioning.
G92: Set part coordinates.
G98: Can cycle start.
G99: Can cycle start.
G192: Set artificial home position.

M00: Program stop.
M01: Optional program stop.

M03: Spindle clockwise.
M04: Spindle Counterclockwise.
M05: Spindle stop.
M06: Tool Change.
M08: Coolant ON
M09: Coolant OFF
M10: Turns ON relay #6
M11: Turns OFF relay #6.
M12: Turns ON relay #7.
M13: Turns OFF relay #7.
M14: Turns ON relay #8.
M15: Turns OFF relay #8.
M21: Changes corner rounding.
M30: Program end.
M35: Tool release.
M36: Tool clamp.
M99: Program repeat.

0.2 Introduction:

The LEVIL 10M control is a combination of hardware and software to make a milling machine work as a ***Numerically Controlled*** machine using servo drives with encoder feedback.

The control consists of:

Chip used is LM-629 from National semiconductors.
- 4 simultaneously controlled axes,
- 16 inputs
- 8 relay outputs, 5 amp. rated
- 1 digital spindle output, 8 bit resolution.
- 1 analog spindle output.
- 1 wheel input, for manual operation.
- PWM and +/- 10 volt, analog servo control.
- 3 amp continuous, 6 amp peak servo amplifier for DC-brush motors (when used in small machines).

The control uses a parallel printer-port to communicate with the host computer. Although any type of computer can be used, the software is

written for a PC-based computer with Windows 95, 98 , Me, Windows NT, 2000 , XP or Vista. Make sure to use a period (not a comma) for decimal point for punctuation standard. The Communication port is USB special serial device attached to the main board.

To install the software, simply copy the program SP10x.exe to the directory of your choice in your hard disk and run it. You also have to install the driver for the serial port, included with software; otherwise, the connection between the PC and the control is impossible. The SP10x.exe program will automatically create a SP10.ini file which contains the parameters of your machine tool and will be located at the same directory. For the LV-300D milling machine, the parameters come already set. However, many end-users may like to make some changes. Be very careful, parameters control the machine tool, some parameters may cause erratic behavior, especially the servo controls.

The SP10x.exe program should automatically determine the port of the computer and the active axis running. For the milling machine X, Y and Z axis should appear automatically if no fourth axis is used. **Do not run more than one SP10x program at the same time** since it can cause a communication conflict between them, usually seen by erratic display of position and intermittent switching of relays. If you do RUN more than one SP10x.exe, it is necessary to RESET the computer, so that the USB driver is reloaded.

NOTE: make a copy of all your files: SP10x.exe, SP10.ini (tells the computer your machine configuration). Better yet, copy the directory.

1 OPERATION:

1.1 Main buttons:

1.1.1- **Reset:** Stops axis motion and returns everything to its steady state position. It DOES NOT turn OFF the power or the servos. You can RESET the control by pressing F1 on the computer keyboard or clicking the **"Reset"** button on the main window.

1.1.2-**Hard Reset:** Produces a total reset of the system. The servo absolute position is lost, and the control is disconnected. HARD RESET is performed when the control becomes erratic or an axis does not want to move for any reason. This is normally performed by turning OFF the control in most machines as well as this one. The HARD RESET is provided mostly for research of the control system and not for normal operator usage.
If the control system experiences erratic behavior often, then something is wrong and should be repaired. In fact, there is no direct access to this command by the operator. To gain access to this command, press COMMAND on the top menu, and then press HARD RESET. A hard reset BUTTON should appear under the **"Reset"** button, and then click the HARD RESET button. Do home positioning after every hard reset.

1.1.3-**Connection:** If the computer detects that everything is OK then it automatically connects the machine ready-relay. This connects the power to the servos. You can Disconnect the machine through the computer pressing Control-D or clicking the mouse on the **"Disconnect"** button on the main window. Similarly, it applies to the Connection; hit Control-C or click the **"Connect"** button. Sometimes, if there is electric noise, a axis may lose its internal parameters thus loosing its position. This is dangerous if the program is run with the axis off position. You must take this axis to home before continuing.
Note: Then LV-300D milling machine has a Red emergency push-button to stop all power to the servos and spindle.

1.1.4-**Axis Motion:** Axis can be moved manually by clicking the mouse at the different axis buttons, or by hitting the keyboard key corresponding to the axis movements. These keys are chosen by parameters on the axis parameters window. Axis can also be moved manually using the pendant wheel (optional). Steps of 0.001, 0.01 and 0.1 mm can be chosen by the switches on the wheel. The **"Wheel"** button is a toggle switch which activates the wheel.

If the axis is moved by pressing buttons, the speed is set by the manual speed window and the distance by the manual step window. If you choose a very low step, say 0.001 mm, and very high speed, say 2000 mm/min, then the axis will move very slow due to the acceleration and deceleration in every step.

1.1.5-**Home Position:** On power-up the control has no idea of the absolute position of the machine, although it will run the programs, it will

not remember its position after a power-down or **Hard Reset**. To remember the absolute machine coordinates, a Home position process should be performed. Simply activate the **"Home Act"** check-box and hit the axis to be homed in. Only do one axis at a time. If you hit the axis in the wrong direction, it will go to the end of travel and stop, if an error is caused, it will disconnect the servos, you must reconnect and activate the **"Home Act"** check-box. Once the home in is performed, hit **"Reset"** and an asterisk will show at the right of the axis position to tell you that home has been performed. Even if the computer is reset or turned off, the absolute position will be maintained by the control as long as the control is ON and no hard errors have occurred.

1.1.6-**Manual Speed:** Is the axis speed when operated manually. To increase the speed press F6, and to decrease speed press F5. You can also click the mouse button.

1.1.7-**Manual Step:** Is the distance traveled each time a movement is commanded by a manual input. To increase step press F8. To decrease step press F7.

1.1.8-**Cutting Feed Percentage:** When running a program, the G01, G02 and G03 feed is multiplied by the percentage of the programmed cutting feed. To increase press F10, to decrease press F9.

1.1.9-**Rapid Feed Percentage:** When running a program, the G00 feed is multiplied by the Rapid Feed %. To increase press F12, to decrease press F11.

1.1.10-**Spindle ON Clockwise**: to turn ON the spindle press <control-F2> at the same time. The speed of the spindle is given on a window above. To change the speed use "< or >" or write the RPM on the spindle window and hit < SPL set>,, activate with the set button. To stop spindle hit the <reset> or <F2> button.

1.1.11-**Spindle ON Counterclockwise**: To turn ON the spindle press <shift-F2> at the same time. The speed of the spindle is given on a window above. To change the speed use "<" or " >" or write the RPM on the spindle window and hit < SPL set>, activate with the set button. To stop spindle hit the <reset> or <F2> button.

1.1.12-**SPL set**: this button is to set the spindle speed. You can change the spindle speed in the spindle window, then press “SPL set” button to set that speed.

1.1.13-**Opt. Stop**: when M01 is used to stop the program, the Opt. Stop check box must be checked for this to happen.

1.1.14-**Show Prog**.: When a program is running the MDI window appears to show the program variables and data. If you don’t want to se how the program is running, uncheck the “Show Prog” check box.

1.1.15-**Show Error**: This check box opens the editor window showing position error history from the opening of the program.

1.1.16-**SPL Interlock**: The SPINDLE Interlock check box stops motion if the program runs in cutting mode (G01, G02, G03, G81 etc) with the spindle OFF. (Spindle must be on for cutting mode).To allow the program to run in cutting mode with the spindle OFF, uncheck the SPL Interlock check box.

1.1.17-**/ Block skip**: if you use “ / “ to skip a block, check the “/ block skip” check box, otherwise the “ / “ will be ignored. Note: use the “ / “ at the beginning of the line, otherwise there will be an error.

1.2 MAIN MENU BAR:

1.2.1-FILE

1.2.1.1 NEW: When a new program is edited, use “SAVE AS” to save, otherwise it will be saved with the last program name, thus erasing the last program data.

1.2.1.2 OPEN: Opens a program file from the disk.

1.2.1.3 SAVE: Saves a program file to a location indicated at the parameters’ list.

1.2.1.4 SAVE AS: Saves a program file with a different path.

1.2.2-WINDOW:

1.2.2.1 TEST: Opens the INPUT-OUTPUT window. You can see how the switches and control chips are behaving.

1.2.2.2 GRAPHICS: Opens the GRAPHICS window, see 1.5 GRAPHICS for operation.

1.2.2.3 ISOMETRIC GRAPH: Opens the ISOMETRIC graph window.

1.2.2.4 Run MDI: Opens the MDI window. Under this window you can see all the details of a running program. You can also run a short list of commands.

1.2.2.5 DIGITIZE: Opens the Digitizing window, see 1.6 for DIGITIZING.

1.2.3-SETUP:

1.2.3.1 OFFSETS: Open the offset and setup window. For setup see 1.4 SETUP. Number can be inputted to change the length of a tool, or change coordinates.

1.2.3.2 Parameters: View and change parameters under this window.

1.2.4-MOVEMENTS: Pressing this buttons makes the axis run manually.

1.1.4.1- X+: Code for keyboard stroke.
1.1.4.2- X-: "
1.1.4.3- Y+: "
1.1.4.4- Y-: "
1.1.4.5- Z+: "
1.1.4.6- Z-: "
1.1.4.7- A+: "
1.1.4.8- A-: "

1.2.5 COMMANDS: Lists the Keyboard codes for a series of commands. Clicking these commands with the mouse, will make it happen.

1.3 PROGRAMMING: The programs can be edited directly into the edit window. Use only capital letters. If different a code from the ones mentioned is used, it will be ignored.

Once the program is written, press <ctrl-R> to run that program.

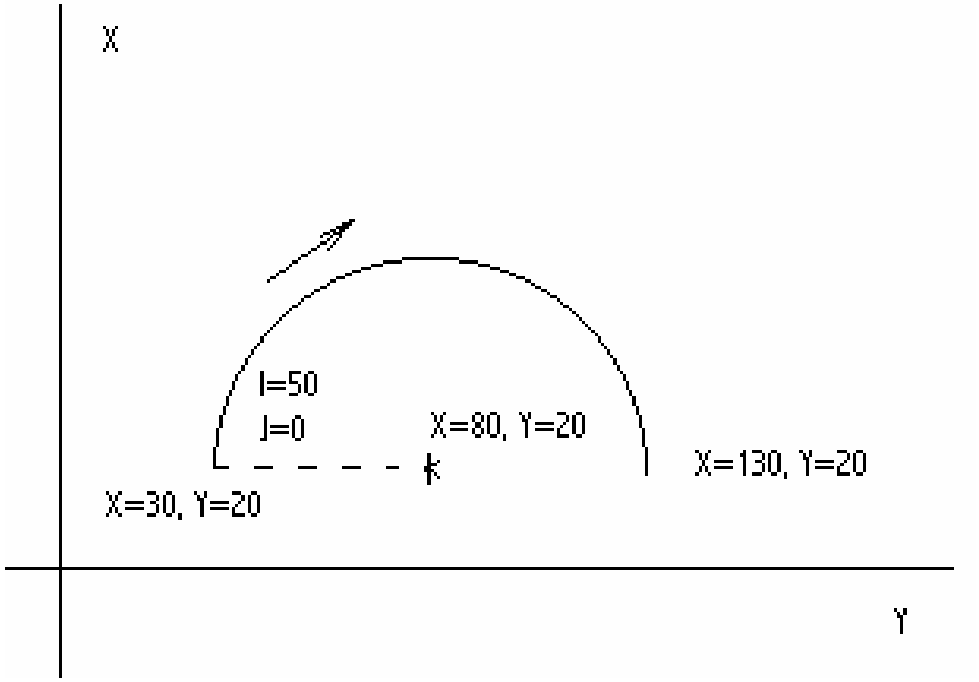
G00: Positioning code for rapid travel.

G01: Positioning code for linear interpolation must have a feed code, or the last commanded feed will be used. Ex.: G01 X20.3 Y1.2 F30.

NOTE: When using G01, G02 or G03 and any other cut function like CAN cycles, it is necessary to have the spindle ON, otherwise the control will; not execute the function. If your application requires that the spindle be OFF during a cutting function, then turn OFF the “spindle interlock” check mark at the main window.

G02: Clockwise Circular interpolation must be accompanied by an I, J, K, X, Y, Z, depending on plane used. If working on the X-Y plane, we want an upper half circle of 50mm radius. With center at X=80, Y=20, then we can write:

```
G00 X30 Y20
G17
G02 X130 I50 J0 F200
```



Note that the Y=20 remains the same. The G17 command tells the control that the circle is in the XY plane. The I and J commands is the distance from the starting point to the center of the circle.

G03: Is the same as G02, except it is counterclockwise.

G04: It is a timer, if you would like the program to wait for a period of time given in milliseconds. Ex.: G04 P2000. The program will idle for 2 seconds.

G10: Skip function. Used similar to G01 except the travel is conditioned to a signal from the touch sensor. The machine will travel to the destination as long as there is no contact signal, once a contact signal is received the machine will stop at that point and continue reading the next line. Ex:
G0X20Y40 (machine travels to point X20, Y40 at fast paste)
G10X30F500 (machine moves to X30 at 500 mm/min until contact)
#7=#41 (the X value is assigned to the general variable #7)
G0X20 (the machine returns to X20)

This sample program finds the edge of a part using a touch sensor.

G17: Establish the XY plane as a working plane for circle and contours.

G18: Establish the XZ plane as a working plane for circle and contours.

G19: Establish the YZ plane as a working plane for circle and contours.

G28: Takes the axis home: Ex. G00G91G28Z0. The program takes the Z axis to its Home position.

G43: Adds a +tool offset in the Z-direction. The amount of the offset is the H command. Example: G43 G00 H8 Z20. The tool will be compensated adding the amount in tool number 8 at the offset window.

G44: Adds a -tool offset in the Z-direction. The amount of the offset is the H command. Example: G43 G00 H8 Z20. The tool will be compensated subtracting the amount in tool number 8 at the offset window.

G49: Cancels the offset compensation. Don't forget to put a Z axis point to make sure it moves to that point.

G53: Machine coordinates. Use this command to take the machine to a known point using the absolute (machine) coordinates. Ej. G00G53X0Y0 The machine will go to home in X and Y.

G54: Changes the coordinate system to G54 mode.
G55: Changes the coordinate system to G55 mode.
G56: Changes the coordinate system to G56 mode.
G57: Changes the coordinate system to G57 mode.
G58: Changes the coordinate system to G58 mode.

G61: Exact stop, in G01 causes the control axes to stop after every point.
G64: Round corners, the control axes does not come to complete stop, thus rounding corners in G01.

G80: Can cycle cancel command. Must be commanded at the end of the can cycles.

G81: Can cycle for drilling. You can perform drilling is a simplified way by writing one line with the variables that describes the hole, then just write the positions. The plane is established by G17, G18 and G19 as is done for circle interpolation. The parameters are:

'R' is the beginning of the hole.
'X', 'Y' and 'Z' are the position and end point of the hole. Depending on plane used.
'F' is the tool feed given in units/min.

Ex:
G17
G98 G81 X25. Y48. Z-5. R2. F100.
X20
X10 Y17
G00 Z30.

This example makes a hole 5 units deep using the XY plane. The holes location are:

X=25, Y=48
X=20, Y=48
X=10, Y=17

G82: Can cycle for drilling. Same as G81, except it has a dwell time at the bottom of the hole. You can perform drilling is a simplified way by writing one line with the variables that describes the hole, then just write the positions. The plane is established by G17, G18 and G19 as is done for circle interpolation. The parameters are:

'R' is the beginning of the hole.
'X', 'Y' and 'Z' are the position and end point of the hole. Depending on plane used.
'F' is the tool feed given in units/min.
'P' is dwell time at the bottom of the hole.

Ex:
G17
G99 G82 X25. Y48. Z-5. R2. P250 F100.
X20
X10 Y17

G80 G00 Z30.

This example makes a hole 5 units deep using the XY plane. The drill will idle 0.250 sec at the bottom of the hole The holes location are:

X=25, Y=48
X=20, Y=48
X=10, Y=17

G83: Can cycle for drilling using chip removal braking cycle. The cycle consists in intermittent feed to brake the chips and then removing them by retreating the tool to the beginning of the drilling cycle. You can perform drilling is a simplified way by writing one line with the variables that describes the hole, then just write the positions. The plane is established by G17, G18 and G19 as is done for circle interpolation. The parameters are:

'R' is the beginning of the hole.
'K' distance to brake chips.
'Q' distance to remove chips.
'X', 'Y' and 'Z' are the position and end point of the hole. Depending on plane used.

'F' is the tool feed given in units/min.

Ex:
G17
G83 X25. Y48. Z-5. R2. Q4. K0.5 F100.
X20
X10 Y17
G80 G00 Z30.

This example makes a hole 5 units deep using the XY plane. The tool feeds in intermittent steps of 0.5, then retracts for small amount to brake the chips. Once the tool reaches 4 units, then retracts to the R position to clear the chips. The holes location are:

X=25, Y=48
X=20, Y=48
X=10, Y=17

Note that at the end of the cycles, a Z30. (safety position) must be commanded or a G80 cycle cancel command.

G90: Absolute positioning, modal. This command tells the control that the axis positioning is given in absolute form. If you write G90X40, the control will position at X=40.

G91: increment positioning, modal. This command tells then control that the axis positioning is given in incremental form from the active point. Ex.

G00G90X40
G91X-100

In this example, the spindle will move to position X=40, then it will go to position X=-60.

G92: Set part Coordinates. Under a program, coordinates can be set by the use of G92 followed by the axis distance to the part. Ej.

G91G28Z0
G92 Z123.45
G90 Z50

In this example the machine moves to Z-home and sets Z-axis to 123.45. The changes are automatically made on the G54-G58 active.

G98: Commands the beginning of a Can cycle. It also tells the control to use the safe Z position between cycles.

G99: Commands the beginning of a Can cycle. It also tells the control to use the R position between cycles.

G192: This commands resets the machine counter to 0 on the axis chosen. This command is usually used to reset the rotary axis when it has done many turns, and it takes a while to return to 0.

G00 A720
G192 A0

In this example the rotary axis A is set to 0 on the machine coordinates.

F (Feed): Is the programmed speed for moving in linear and circular interpolation.

H: is the tool offset compensation in the offset window.

M00: Stops the program at that point, works as a program halt.
The program continues once you hit Program Run again.

M01: Stops the program at that point if the optional stop check is checked, works as a program halt.
The program continues once you hit Program Run again.

M03: Turns the spindle forward relay (out 4) ON, and the spindle reverse relay (out 5) OFF.

M04: Turns the spindle reverse relay (out 5) ON, and the spindle forward relay (out 4) OFF.

M05: Turns OFF both spindle relays (out 4 and 5).

M06: Tool change. This command causes a subroutine to be loaded in the program running. This subroutine is usually used to change the tool. The spindle tool is shown on a window in the main program. If you are starting the software, and the spindle tool reads '0', the spindle should be physically empty, otherwise a collision is imminent when M06 is commanded. There is an internal subroutine already programmed for the LV300 series machine. The spindle goes to Z-home, moves to the empty pocket, drops the tool, goes to Z-home, moves to the designated tool (Tx) and picks up the tool.

To designate the position of the tools (T1-T10) use the parameter values in the parameter table. Home is use for the first tool (T1), Put the machine position in mm for T1. The increment is use for the second tool (T2). Put the distance between T1 and T2 (use mm for this). The software calculates the rest of the tools. Make sure that they all align with the same distance.

M08: Turns ON the coolant relay (out 3).

M09: Turns OFF the coolant relay (out 3).

M10: Turns ON relay #6. Do not use this command if tool unclamp is used. This is the same relay as used in tool unclamp. NOTE: M10 is a direct command and no checks are performed. If the spindle is rotating and the machine has unclamping capabilities, NO M10 code should be performed; otherwise the tool will be release rotating. You then will say: !@#$%^&&*.

M11: Turns OFF relay #6.

M12: Turns ON relay #7.

M13: Turns OFF relay #7.

M13: Turns ON relay #8.

M14: Turns OFF relay #8.

M21: Changes corner rounding. Corner rounding is set at parameter 'Step Constant'. To modify this you can program an M21 Pxx.x where xx.x is the amount to multiply the 'Step Constan'.

ie. M21 P0.5
in this sample corner rounding is reduce by ½.

M30: It commands a control reset, program is terminated, servos go to stop, and coolant and spindle are shut off.

M35: Tool release. It works activating relay No. 6 after verifying the spindle rotation. Do not activate this command during spindle rotation.

M36: Tool Clamp.

M98: repeats the program from the beginning if the value 'common Z' in Offsets is greater than 0. Then amount of 'P' will be subtracted from 'common Z' to decrease until the value of 'common Z' = 0.

This routine is used for repeating the program at different levels until Z level=0 is reached.

Ex: G00 Z50
X0 Y0
G01 Z-7.5
X100 F200
G00 Z50
M98 P-1.0

In this sample the offset position in 'common Z' should be set to 6.5, it will decrease automatically as the program reaches the M98 command.

The last 0.5 is subtracted rather than 1.0, to make the end result Z=0. This way you can plan for the last pass to be a finish pass.

M99: repeats the program from the beginning.

1.3.2-Macro variables

The control has a way to make primitive macro functions. It is possible to write variables to axis to do simple tasks like automatic tool measure.

You can assign a variable to an axis like 'X#4' were #4 is a variable of changing values.

The following is a table to look for the value behind the variable:
#1 - #10 is a set of generic variables. Use them for any purpose.
#11- #20 are the length offsets. Use to change the Z offset values. #11 corresponds to tool offset No1. #20 corresponds to Tool offset No10.
#21- #30 are the radius offsets were #21 corresponds to radius offset No. 1.
#41-#46 Work coordinates values.
#46-#49 Common offset.
#51-#54 G54 offset.
#56-#59 G55 offset.
#61-#64 G56 offset.
#66-#54 G69 offset.
#71-#54 G74 offset.

1.3.6-**MDI window operation**: The MDI window appears automatically when a program is running, or is called by the operator by clicking Window>RunMDI. The MDI window shows the program and its variables as it runs. It can be used to run short programs for centering, turning ON various relays, etc.

If you like to run the program at a point different than the beginning, do an N search: write the N number that you like to search for at the upper right corner of the MDI window, then check the <Check to search> check box, the search begins here, if the N number is found the program will start loading from that point. To run the program press <ctrl-R>, execute, or run program button.

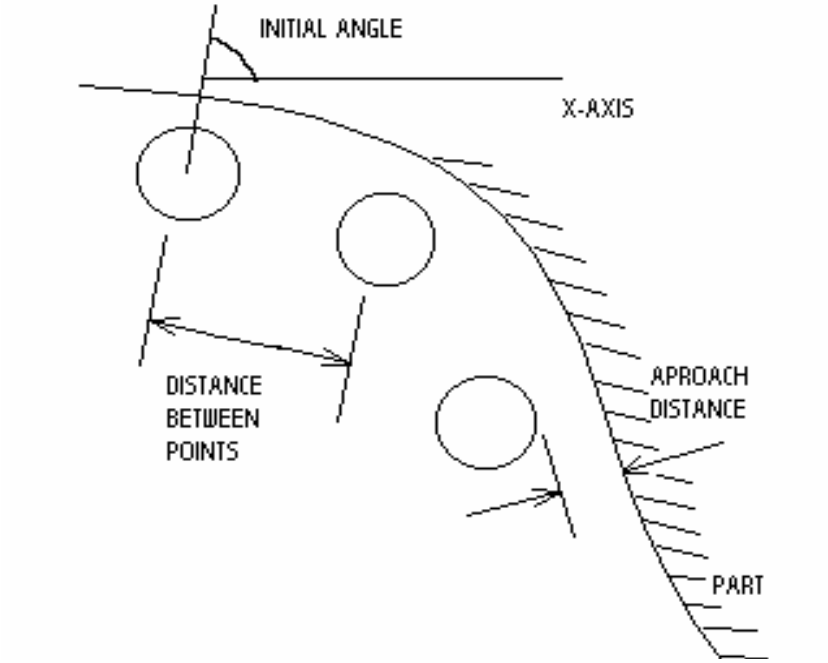
1.4 SETTING UP: To be able to use the machine with the axis set to ZERO at the correct position of the part, do the following: Choose a set of coordinates (G54, G55 or G56). Move the axis to a known position of the

part, perhaps X=20 for G54. Call the offset window by pressing the INSERT key on the Keyboard or use the main menu clicking SET UP then OFFSET. Write the value at the VALUE window ("20." in this case), write the axis at the AXIS window ("X" in this case) and press MEASURE. This will change the X axis offset window in the G54 coordinated system to make the X-axis = 20.0 in the work coordinate system. It sounds complicated, but is very simple. The number written in the X window of the G54 coordinate system is the distance from the home position to the 0 of your part. The Home position or machine X=0 point plus the distance to your part gives the correct position for the counter of the X-axis work position.

1.5 GRAPHICS: Graphics are shown in 3 planes and isometric. To see the isometric window click <Window><Isometric Graph>. To see the 3 planes window press <Window><GraphicsXYZ>. To see the program click <Show prog>, after it runs for a while, press <Stop Prog> then <Auto Scale>. This fits the window size. Press <Erase> to clear the window and <Show Prog>. You can also set the window size manually by inputting axis limits and press <Center Scale> or inputting centers and scale and press <borders>.

1.6 DIGITIZING: This is a way we can translate physical shape to computer file. There are different ways to follow a contour and output the numbers. This control uses a touch type of sensor; there is an indicator in the main window showing the sensor output. This means that the sensor has to separate from the surface every time a new point is needed. It is not the fastest way to digitize, but it does the job, and many times a lot cheaper. The way the sensor follows the part determines the number of variables to be used. Since there are different types of digitizing available: CONTOUR, SURFACE and MULTIPLE CONTOUR, the required number of variables will change. We will start with the simplest type offered:

1.6.1-CONTOUR: Is designed to follow an unknown contour on a given plane. It will go around until the first point position is reached. Click the CONTOUR check box in the digitizing window. To start the process we must specify a few variables:



1.6.1.1-INITIAL ANGLE: Is the direction of travel at the start of a contour. Since the direction is given with an angle value, being zero the positive X-axis, going counterclockwise if you use the XY and XZ plane. For the YZ plane, a positive Y-axis is zero.

1.6.1.2-DISTANCE BETWEEN POINTS: Is the distance parallel to the contour that the sensor will travel to get the next point. To choose this parameter, a careful consideration must be made of precision, time for digitizing, radius of sensor and corner angles. Parts with sharp angles should have this parameter at least 0.2 times the diameter of the sensor; otherwise, it will be impossible to go around sharp corners.

1.6.1.3-APROACH DISTANCE: Is the distance that the sensor will separate from the surface to go to the next point. If the contour is smooth with small change in angles, then a short distance is effective. You can start with a distance similar to the distance between points and then change as you get experience. The shorter the APROACH DISTANCE the faster the digitizing process.

1.6.1.4-PROVE FEED: Is the speed that the sensor approaches the contour wall. The faster the speed, the lesser precision is achieved; of course, the computer speed and the number of programs running at the same time will affect the precision. Normally when digitizing, only the SP10.EXE or the SP9.EXE (DOS version) program should be running.

1.6.2-**SURFACE:** This type of digitizing offers the operator the possibility of doing a series of contour lines enclosed by a rectangular limit.

It is offered for the XY-plane only. You can choose if the lines follow the X-axis or the Y-Axis.

1.6.3-**MULTIPLE CONTOUR:** It is a series of contour lines done in the XY plane with decrement in the Z-axis. Position the sensor at the deepest point of the section you want to do, set the parameters and press start.

The procedure will yield a file in different formats. Each file will have a name given at the PROGRAM NAME window followed by a dot and a file type extension. The different types of file are:

a. NC Type of Format: Will reproduce the trajectory followed by the digitizing sensor in an NC program. (G-code program). To get an NC file, just click the **CheckNC** file check box in the digitizing window.

b. MCL Type of Format: Will reproduce a spline curve to be run on SmartCam software. This format is run under the custom macro execute routines. To get an MCL file, just click the **CheckMCL** file check box in the digitizing window.

c. DXF Type of Format: Will reproduce a DXF file to be loaded in most cad-cam systems. Sometimes the newer cad systems rejects this old format DXF, if this is so, just copy the heading of any program containing a line and paste it on top of the DXF digitized file. To get a DXF file, just click the **CheckDXF** file check box in the digitizing window.

d. TXT Type of Format: Will reproduce a set of numbers corresponding to the X, Y and Z axis every time there is a contact. This is use for Pro Eng. Software to generate a cloud of points, thus generating a surface. To get a TXT file, just click the CheckTXT file check box in the digitizing window.

To be able to digitize, first some information has to be given so that the sensor can follow the path.

2 PARAMETERS:

There are different kinds of parameters: GENERAL, General INPUTS and AXIS. The GENERAL parameters control the machine applying their contents to all axis and input output ports. The General INPUTS is the same as GENERAL but are in another window. The AXIS parameters are specific for each axis and are in windows at the write of the GENEARAL parameters. To change the parameters you need a password “1588”.

2.1-GENERAL parameters:

2.1.1**Program Name:** Is the path where the name of the NC program is located.

2.1.2 **Lubrication Time**: Is the length of time in seconds that the oil pump is turned on every time there is an oiling cycle.

2.1.3 **Lubrication Distance:** Is the sum of the distance traveled by all axes, at the end of which a lubrication pulse is given.

2.1.4 **Time between Lubes:** In addition to distance traveled, the machine will lubricate every X seconds.

2.1.5 **Decimal Places:** Is the amount of decimals visible on screen.

2.1.6 **Acceleration:** Is acceleration of the servo motors, the more the power of the motors, the more acceleration that can be used. Also the more mass of the axis to be moved, the less acceleration allowed; since it's a pulse/sec/sec.
It depends on the encoders used on the servos.

2.1.7 **Max Spindle Speed:** It calibrates the Spindle Output electronics to the physical limit of the spindle motor.

2.1.8 **Max G00 Velocity:** Is the velocity when G00 travel is commanded at 100%. Sometimes this velocity combined with acceleration determines the maximum performance of your machine. If fast acceleration is required, but G00 velocity is not important, then a combination between the two is possible to get better performance out of your machine.

2.1.9 **G83 Cutting Distance**: Is the retract distance when G83 cycle is used.

2.1.10 **Wheel Division, Sign:** Is the direction and quantity of movement given by the wheel. Since it's a division, a zero will yield an error.

2.1.11 **Key Port:** 08 or 12. This is the input port for the pendant keyboard. Some controls use Port 08, others use Port 12.

2.1.12 **Arc Constant:** Is a constant to determine the sections of the arc when in G02 or G03 mode. The smaller the constant, the more precise the arc section limiting the maximum cutting speed.

2.1.13 **Step Constant:** Similar to the arc constant, but use in the G01 mode.

2.1.14 **Max Step Contouring**: Following a series of G01 commands, the maximum step allowed to go to the next line without stopping.

2.2-General INPUT parameters: There parameter are use to control the keyboard input. There are 6 inputs that can be combined to generate 64 different combinations of inputs. To choose the input code, simply press a key on the pendant keyboard, a code will show on the message window. Put this code at the parameter window that you want to use. The different functions that can be accessed using the pendant keyboard are:

Reset
Spindle Forward or Output 4 ON
Spindle Reverse or Output 5 ON
Spindle Stop or Outputs 4 and 5 OFF
Manual Velocity Increment
Manual Velocity Decrement
Manual Step Increment
Manual Step Decrement
Wheel input ON
Wheel input OFF
RUN Program
STOP Program
Coolant ON or Output 2 ON
Coolant OFF or Output 2 OFF
Home Positioning ON
Home Positioning OFF
Spindle speed increment
Spindle speed Decrement
Tool Unclamp
Tool Clamp

Output 7 ON
Output 7 OFF
Output 8 ON
Output 8 OFF
Connect or Output 1 ON
Disconnect or ALL outputs OFF
Feed Increment (G01, G02, G03 feed %)
Feed Decrement (G01, G02, G03 feed %)
Rapid Increment (G00 feed %)
Rapid Decrement (G00 feed %)
Door Interlock

For AXIS movements using the pendant keyboard, the parameter is inside the AXIS parameters and the inputs are similar.

The intput to the machine is through a DB-9 male connector with the following pinout:

1- 5 volt
2- input
3- input
5- GND
6- input
7- input
8- input
9- input

to get a reading the input cable connects to GND.

2.3-AXIS Parameters: To view or change, press one of the four AXES small windows located at the right of the parameters window.

2.3.1 Key direction: Is the direction of the axis movement, input 1 or -1 to change direction.

2.3.2 Axis Direction: Is the sign of the axis, input 1 or -1 to change the sign.

2.3.3 Home Direction: Is the direction of travel once the axis has hit the home switch. Input 1 or -1 to change direction.

2.3.4 Home Velocity: Is the velocity of travel once the home switch is pressed.

2.3.5 In Position Error: Is the maximum allowable error while the servo is standing still.

2.3.6 Moving Error: Is the maximum error allowed, the control will shut down the servo power and other peripherals.

2.3.7 Number of Pulses per Unit: This is the number of pulses that the encoder produces per unit of travel. Example: A 5mm pitch ball screw and a 1000 line encoder connected 1:1. A 1000 pulses/rev produces a 4000 counts/rev, so we divide 4000 by 5000 microns giving 800 counts/mm.

The next three parameters relate to the equation that governs the servo movements. Since this varies for every machine type, it is difficult to set the right constant. The equation looks like this: Ev = (Kp * Position error) – (Kv * Velocity error) + Integral(Ki * Position error).

2.3.8 Position Constant: Kp is the constant that controls the amplification of the position error. The bigger the error, the bigger the voltage needed to compensate this error. If too much voltage is applied, the system will become unstable.

2.3.9 Velocity Constant: Kv is the constant of the first derivative destined to allow a higher Kp maintaining a stable system.

2.3.10 Integral Constant: Ki is the correction for the error produced by applying a steady force to the servo.

As a general rule, to set this parameter, first set them to 0, then increase Kp slowly until the servo becomes unstable, then set Kv = 10 * Kp, this should make the servo stable again. Redo this until a vibration of high frequency is detected. Go down a little. Then increase the Ki until vibration occurs, then go down a little.

2.3.11 Positive limit: Is the software limit for the axis.

2.3.12 Negative limit: Is the software limit for the axis.

2.3.13 Backlash: Is the compensation for the axis mechanical play.

2.3.14 Identification Letter: Is the letter that will show on the screen for that axis.

2.3.15 Key Code FWD: Is the code for the axis forward movement using the pendant box.

2.3.16 Key Code REV: Is the code for the axis reverse movement using the pendant box.

2.3.17 Keyboard Key FWD: Is the key in the computer keyboard to create a forward movement of the axis.

2.3.18 Key REV: Is the key in the computer keyboard to create a reverse movement of the axis.

2.3.19 Convert: Is the amount use to change the axis unit system. If the machine is setup in mm, '1', then use this parameter to change the axis to INCH system. Put 25.4 in this axis. All axis must have the parameter written. Usually the forth axis should be left with '1' since is set-up as a 360 degree turn.

3 Troubleshooting:

3.1 No sign of life: Check for connections, hit the connect-Disconnect button on the main screen, you should hear a relay click. If you don't, there is no connection between the control board and the computer. Check for burned fuse or lack of power on the main board.

3.2 Control is crazy: relay turn ON and OFF in random sequence. The axis position gives erratic numbers. Probably you are running more then one program "SP10x.EXE" at the same time. Close all windows and restart program.

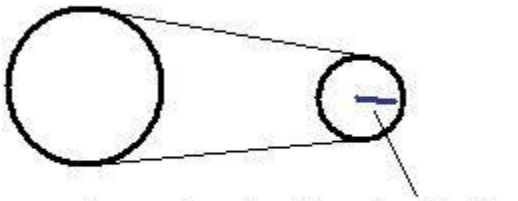
3.3 The control connects, the relays work (coolant, spindle, etc) but the axis don't work. Probably you have a parallel port in unidirectional mode or ECP. You must reprogram the BIOS with bidirectional or EPP mode.

3.4 Encoder works, but motors don't: Connect the control and pull the emergency button if installed, the main power could be OFF.

3.5 One or more axis don't move: The control chip for that axis is frozen: Press the CONNECT button once or twice to reload the parameters. If that fails, turn control OFF and ON, do a home return. If you don't like to turn OFF the machine, you may use the "Hard Reset" feature to Reset the chip. This is equivalent to turn OFF the control. Of course a home return is a must after a "Hard Reset".

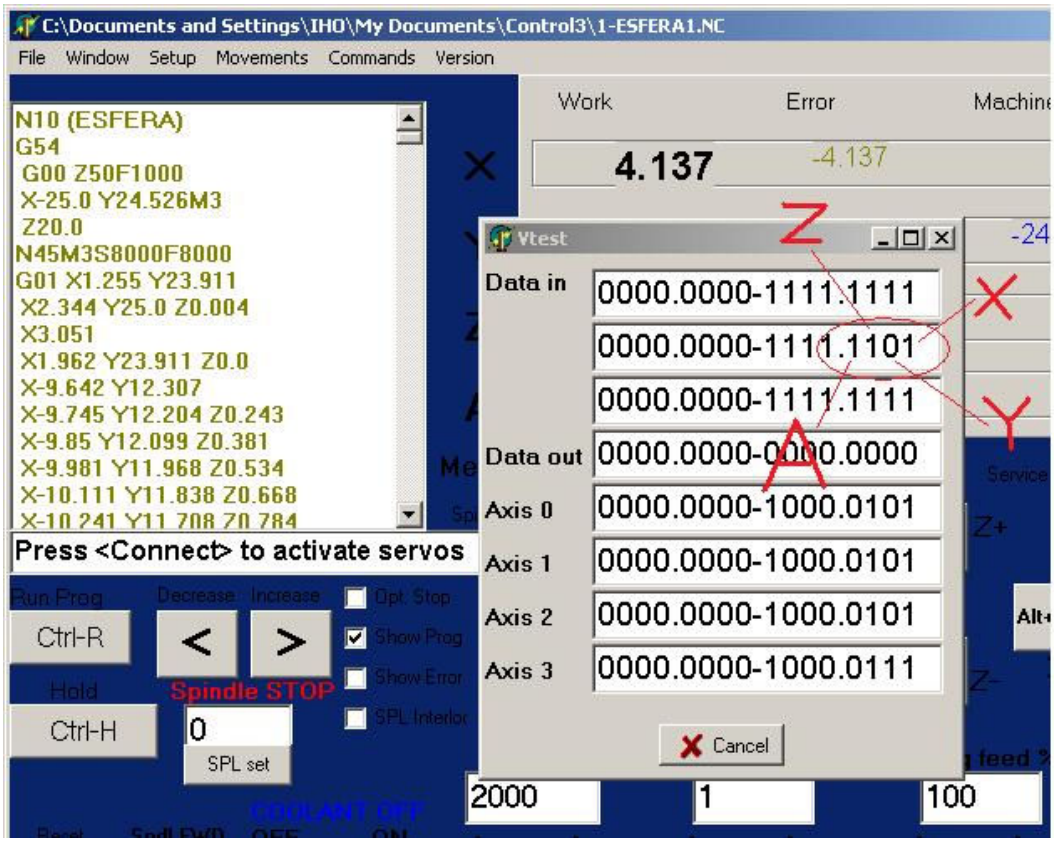
3.6 Home position varies, two different home positions are obtained. Usually this is due to an index pulse too close to the home switch trigger. The following procedure is done to eliminate this possibility:

Take the axis 'HOME' and mark a line away from the big pulley, this will tell us where the index pulse is:

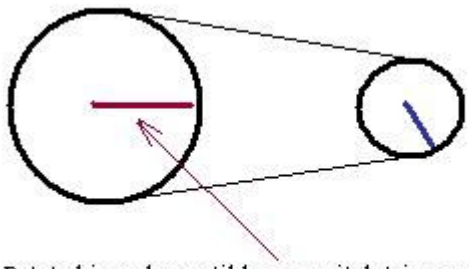


Go home and mark position of small pulley away from big puley

Press the emergency button or disconnect the servo power so that the blue screen shows, click 'Window' and 'Test' to bring out the Input-output window:

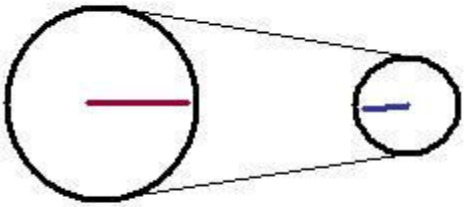


Rotate manually the big pulley towards the home switch until the the number of the axis being rotated changes from ‘1’ to ‘0’. This is the point where the home switch triggers. Mark this position with a line pointing to the small pulley.



Rotate big puley until home switch triggers and mark that position on the big puley with a line toward small pulley

Since we want the index pulse to be away from the triggering point of the switch, loosen the servo and align the two lines to each other while the big pulley is at the triggering point of the home switch.



At the triggering point of the home switch, Loosen the servo and allign the marks of the puleys towards each other

Pull the belt just snug, no very tight but tight enough that there is no play. Now the index pulse is away from the switch.