AUTO-CODE Wire EDM Programming

This chapter explains the various components found in the standard wire EDM cutting interface for AUTO-CODE. Note that not every wire EDM interface will use all of the elements in the interface. In many custom interface cases, both the dialog boxes and basic operation of the software will be different. The descriptions in this chapter should help you interpret custom changes to the standard interface.

Wire EDM programming in AUTO-CODE works best with polylines in the AutoCAD drawing. Polylines are continuous line and arc sequences that can be used to represent the NC/CNC program path. You can break the polylines wherever needed for tabs and breaker points in manufacturing or you can continuous cut complete closed sequences of geometry. AutoCAD provides a host of commands for creating and manipulating polylines.

The Main Dialog Box

From the main dialog box, titled AUTO-CODE Wire EDM Programming, you make the decisions as to what you want the machine tool to do next. The current machine tool status is shown in the upper left corner. Below the status area are buttons to access the detail settings dialog box, view the drawing, and move the cutter to a new position. Radio buttons located in the middle left provide a quick jump into the AUTO-CODE command options as shown in the icon options to the immediate right of the radio buttons. The Machine controls menu is a programmable list of miscellaneous controls you can access to provide support for the type of work you need to do. Buttons at the bottom provide access to other features in the system as described on the next few pages.
**Burn ON/OFF** – Indicates if the wire cutting or burning is currently on or off. This setting corresponds with the G0 or G1 modes of the machine tool.

**Wire position** – The current tool position of the wire in machine coordinates. These are not the AutoCAD coordinates of the machine tool; they are the adjusted coordinates based on the origin point supplied when AUTO-CODE was started.

**Settings** – Access the settings dialog box in which wire EDM variables, offsets, feed rates, and other parameters associated with setting up the machine tool can be established. When programming a wire EDM, this button is generally selected first in order to set up power levels, feed rates and offsets unless they are always used as supplied in the default setting values. More information about the settings, as well as how to establish the defaults, can be found in this chapter.

**Move** – Reposition the tool to a new spot. This option is used when you need to move the tool around some fixture or between parts and there is no geometry to select. When making rapid positioning moves, make sure the burn on toggle has been turned off.

**Show Path** – The movement sequence of the NC/CNC path will be shown on the graphics screen. Arrows indicating the path will be drawn for each step of the program. Select a line or arc object that is part of the NC/CNC path and the sequence will continue from the selection. The following prompts will be shown when the Show Path button is selected. The distance and location for each move in both absolute and incremental values will be presented.

\[
\text{Move to } X, Y \\
dX = x.xxx \\ dY = y.yyy \\ Distance=d.dddd
\]

Press the Enter key to continue through the sequence, as <Next> is the default option. Type Z for zoom control options, R to refresh the display showing arrows for only the current move, D when done viewing, and B to back up through the movement sequences.

Show path is intended for use when viewing a program already constructed or mostly constructed. If your attention gets drawn away from AUTO-CODE, the show path operation can help remind you what you were doing.

**View** – Clicking this button will cause the main dialog box to disappear allowing you to view the drawing behind it. As in all AutoCAD commands, transparent Zoom and Pan can be used to move the display to another view while the drawing is visible.

Radio buttons toggle the icon display to the immediate right between the options available for wire EDM cutting. You can also directly access the AUTO-CODE command by selecting the button below the radio box.
**Inside Cuts** – Displays the icons for starting up AUTO-CODE with automatic leaders towards the inside of closed geometry objects. These cuts are typically applied to interior cuts of the part being made.

**Outside Cuts** – Displays the icons for starting AUTO-CODE with automatic leaders generated towards the outside of closed geometry objects. These cuts are typically done on the outer perimeter of the part being made.

**Machine controls** – Variable content list menu containing machine controls that can be used to implement various features of your particular machine tool. This list may be quite extensive or very sparse depending on the needs of your programming. Typical applications include switching between different cutting modes and power levels as well as generating the output of the final machine codes needed to terminate a program or subroutine.

**Generate NC** – The Generate NC button is usually the last button clicked when writing programs inside of AUTO-CODE. A dialog box will appear asking what file name to use and then the translator will start. The translator takes the tool path information along with the settings information and creates an NC/CNC output file.

**Cancel** – Will exit AUTO-CODE program and return you to the AutoCAD command prompt. AUTO-CODE updates will remain in the drawing. This option is used to add new geometry or perform other edits in AutoCAD and then return to AUTO-CODE. To return to AUTO-CODE after exit through the Cancel button, type the command NCG.

**Help** – Click this button to access the AUTO-CODE online help files. Several HLP files are supplied with AUTO-CODE and this button will start the general help found in the AUTOCODE.HLP file.

**Undo** – Undoes the last AUTO-CODE command sequence initiated. Click the mouse “Pick” button to continue the undo operation or press the Enter key to return to the AUTO-CODE main dialog box. Movement mistakes or control set up mistakes can be corrected immediately if caught during the programming session.

**Edit Path** – When mistakes happen in the tool path, you can correct them using the options in the edit path dialog box, Path Edit Options. This can be used to define new sequences of movement in your program as well as to adjust existing paths and points.

**Move Point** – To move just a control point (or group of points) located at the end point of some geometry, use the move point option. All control points will be shown. Make a selection near the control point of interest and then show the new location. All associated control points should move along with it.
**Change path** – Modify an existing path by adding new points into the middle. This option is used to program avoidance moves between parts in large machines.

**Delete pt.** – Remove control points where selected.

**Insert mode ON** – Insert complete new sequences of code into the current program at any point. This feature is helpful when adding new cuts to an older program or when adding additional tool operations into the middle of an existing program.

**Insert mode off** – Turn off current level of insert mode. Tool point will return to location prior to starting the insert mode. You can have over 32,000 insert mode toggles in an AUTO-CODE program drawing.

**Edit CFG** – Start the AUTO-CODE translation control module described in the CUSTOM.HLP file. Translation control files (CFG) are used when generating the NC output.

**What’s a control point?**
Control points are the data objects of AUTO-CODE inside a drawing. When inserted into a drawing they control how the NC/CNC program will look, what tooling it utilizes, and at what speeds. Control points contain information for tool movements as well as tooling setups and machine controls.

In AutoCAD operator terminology, a control point is a block containing two attributes. Do not change the structure of this block or you will cause AUTO-CODE to fail.

Control points can be embedded inside other AutoCAD blocks to create clusters of control points. You can re-use the blocks created in this manner over and over again for standard code sequences such as tool changes or advanced machine control sequences such as table changes.

For those individuals customizing AUTO-CODE, a great deal of information regarding control points can be found in the CUSTOM.HLP file in the AUTO-CODE program directory.
The Settings Dialog Box

The values in the AUTO-CODE Wire EDM Settings dialog box control the machine tool standard elements, such as the feed rate and cutter compensation values. These settings can be established by setting the appropriate constants in the custom interface AutoLISP file (see …\POSTS\WIRE.LSP for example).

The elements found in the settings dialog box may vary from one machine tool interface to the next. Custom interfaces will often add or subtract data items to meet their requirements. The default laser cutter dialog box control information is described in this document.

**XY Feed rate** – The cutting speed of the wire typically expressed in Inches Per Minute [IPM] or the metric equivalents. The feed rate value supplied is output in the machine code as directed by the interface and AUTO-CODE is not concerned with the units used.

**Wire Feed** – The feed rate or travel speed of the wire itself. This is not the cutting speed but is the speed of the wire moving from spindle to spindle. Not all wire EDM interfaces use this value as the controller may not allow the speed of the travel to vary under program control.

**Power setting** – The power setting for the wire EDM cutter. The machine tools interface patterns and AutoLISP utilities use this value to construct the NC/CNC output. If your machine does not use a programmed power setting, then disregard this setting.

**Cutter Compensation** left and right is controlled in this section of the settings dialog box. Cutter compensation is when the machine tool controller will offset the tool path to the left or right as requested. The amount of the offset is
based on a value stored inside the machine tool. The advantage of using cutter compensation is that as wire cut sizes change due to varying materials and conditions, the offsets can be adjusted as well without changing the original program code.

Cutter compensation command sequences are defined using patterns inside AUTO-CODE. Patterns are stored in ASCII text files located in the \POSTS\ sub-directory and are documented in the CUSTOM.HLP file.

**Left** – Cutter compensation left pattern (^CCL) is run when the settings dialog box closes.

**Right** – Cutter compensation right pattern (^CCR) is run when the settings dialog box closes.

**None** – If cutter compensation has been on, then selecting this option will cause the cutter compensation cancel pattern (^CCOFF) to be run.

**Offset Register** – The offset value or register to use when activating cutter compensation. The usage of this value will depend on the machine tool interface. Typically this value is the register in the controller that contains the size of the tool. For example, the CNC statement G42D2 might be the sequence to activate cutter compensation to the right using the offset value in register 2 of the CNC controller.

**Dwell time** – Time to delay when switching cutter modes, waiting for tension on the wire, or any other dwell time. This value is used by the machine tool interface as required and may not be active for all interfaces.

**Taper Angle** – Angle value to be supplied to the output NC file as the taper setting. This angle is not used inside AUTO-CODE and will simply be passed through as typed.

**Taper On or Off** – Enables taper cutting options when selected. Taper cutting can be mixed with cutter compensation and other controls as needed in the interface. Associated patterns in the PAT file will be run when the taper state is changed. A pattern will be run when the taper state is changed. The pattern run will depend on the cutter compensation setting and taper state. When taper is turned on and cutter compensation is set to left, the pattern (^TPRL1) will be run. The name of the pattern that is run is the combination of the letters “TPR” followed by either an “L” for cutter compensation left, “R” for cutter compensation right, and “O” for cutter compensation off. If the taper is being turned on, the digit “1” (one) is added to the end of the pattern name, otherwise a zero (“0”) digit will be appended.

**Water ON and Water OFF** – These radio buttons are used to indicate the status of the water flow. If the controller does not support program control of the water (or dielectric fluid) flow, then these buttons will be disabled. The patterns (^WF0) or (^WF1) will be run when these options are changed. (^WF0) will be run when the water flow is turned off. (^WF1) will run when the water flow is turned on.

**Diameter** – The diameter of the wire cut that is used when graphically showing the wire movement while cutter compensation and tool animation are both active. When cutter compensation is used in programming, the offset value or register edit box is used for the value output to the machine code. This value is the true cut size and is not normally output to the machine code.
**Object snap** – The two choices for **End** and **Center** point provided as toggles. When selected, the object snaps will be enabled for point selections. These are not global settings for AutoCAD. They are used strictly for AUTO-CODE point selections.

**Origin name** – Block name of the origin figure that marks where the machine’s zero point is located.

**Origin scale** – The scale factor at which the origin block is inserted.

**OK** – Clicking the OK button results in AUTO-CODE taking the settings established and placing them in the appropriate global variables for the application. Additionally, the patterns for cutter compensation will run if the state has changed while the dialog box was displayed.

**Cancel** – Returns to the AUTO-CODE main dialog and does not set any of the global variables that may have been changed in the dialog box.

**Help** – Clicking the help button will start the Windows Help system with information about the settings dialog box contents.

---

**The AUTO-CODE Dialog Box**

Selecting the AUTO-CODE button from the main dialog box will present the AUTO-CODE dialog box. Another way to start the AUTO-CODE dialog box is to pick one of the icons representing the automatic leader options from the main dialog box. The AUTO-CODE dialog box is the primary tool used to define the cutting path. The recommended procedure is to draw the objects to be cut as polylines and then apply leaders automatically in the AUTO-CODE system. Leaders are explained in detail at the end of this chapter.

**Pick entities** – Pick entities allows you to select the objects to be converted into NC/CNC program paths. You can use crossing and inclusion windows to select the objects. When the leader features are enabled, the objects selected that support leaders will have them attached automatically. This is the most common method of programming in AUTO-CODE.

**Track entities** – The find path option is used when you have a series of line and arc objects that are connected geometrically. This option is slower and is easily confused when presented with overlaying lines and arcs. It works best with isolated geometry.

**Nearest X-Y** – The nearest X-Y flag activates the automatic routing feature in AUTO-CODE. When on, the system will locate the next geometry from your selection set based on the nearest end point found.
**Allow reverse** – Check this toggle when doing centerline programming so that both end points of an open drawing object are considered. When not checked, only the starting point of the object will be used in the Nearest X-Y routing consideration.

**Auto-offset** – When cutter compensation is not available (that is, no patterns exist for CCL, CCR, and CCO), the automatic offsets can be enabled in an interface (by setting a variable in the LSP file for the interface). This will force AUTO-CODE to place an offset object to the inside or outside of closed geometry. Leaders will then be attached to the offset objects instead of the selected objects.

**Tolerance** – The tolerance setting is used to indicate how far to search around the end points for connecting objects.

**Leader settings** – Button to move you to the leaders dialog box to select the type of leader to use as well as to define the parameters to employ. See the section on using leaders later in this chapter for more details.

**Placement** – Leader placement, select either inside or outside. See leaders in next section.

**Direction** – Cut direction around the object selected. Select either CW (Clockwise) or CCW (counterclockwise). See leaders in next section.

**OK** – Clicking OK will activate the object selection loop. Select the entities representing the tool path. When selecting the objects, the leader lines or arcs will be attached at the location they were selected. A group of objects may be selected by first typing a “C” or “W” to start the selection of a crossing or inclusion window respectively. After typing “C” or “W”, the system will ask for two points defining the boundary of the window. All objects found inside or crossing the window (“C”) or just inside the window (“W”) will be included in the object selection set. Leaders will be attached as described in the next section.

**Cancel** – Returns control to the main dialog box and does not set any of the global variables that may have been changed in the settings dialog box.

**Help** – Displays basic help information about the AUTO-CODE command options.
Using Automatic Leaders

Closed objects such as polylines and circles can have leaders attached to them automatically while the AUTO-CODE command is running. Leaders are objects (lines and arcs) added to the drawing at specific locations where they are attached to various existing geometry objects selected by the operator. The leaders permit the tool to contour after already starting into the cutting cycle and getting up to cutting speed, thereby not marring the part surface or edge.

When selecting objects one at a time, the leaders are generated at the point where the object is picked.

If the windowing options are used to describe a rectangular selection area, the leaders are generated at the top of circles and at the first linear segment encountered in polylines.

Using the Automatic Leader Feature

First select Arcs, Lines, or Corner, for the style of leader. This selection can be made by picking the line or arc lead in icons from the main dialog box for arcs and lines.

Next, select placement as either Inside or Outside the polyline shape. Again, this determination can be made for arc and line leaders by selecting the appropriate icon from the AUTO-CODE main dialog box.

Next, select direction as either CW - clockwise or CCW - counterclockwise. This is the direction of cut around the part and is generally related to whether or not you have cutter compensation engaged and if the cut represents an outside or inside edge. The direction is also controllable from the icon menu level by selecting the type of cut, leader type, and direction graphically. If direction is important, then the objects should be selected one at a time so that it may be controlled independently.

Finally, edit the length and angle if needed. The length and angle define the geometry of the leader. For line leaders, the length is the length of the line and the angle is the included angle from one leader to the other. The default angle is 90 degrees. In some applications it may be appropriate to use larger angles of separation to obtain a closer tangency to the edge from the leader.

When working with arc leaders, the length of the leader is the radius of the arc lead in and the angle is the included angle of the arcs.
Open polylines, lines, and arcs will NOT have leaders attached automatically. Use the AutoCAD PEDIT command to join lines and arcs together into polylines before running the AUTO-CODE command for automatic leaders to be added to the objects.

**Cut Direction**
The direction in which the polyline is cut is determined by a number of factors. The CW/CCW selection is only applied to closed objects that will have leaders attached. The actual direction can be changed when selecting the objects one at a time.

Open objects will be converted to tool paths in the order they are defined in the system unless the nearest X-Y flag is enabled. In that case, AUTO-CODE will find the nearest endpoint and start cutting.

The pilot hole option allows you to define a starting location for the leader. A straight line cut will be made from the pilot hole location to the start of the leader.

The example shown here is for an arc leader with a setup similar to the dialog box below. The pilot hole option has been selected along with the tangent leader into the arc options. The cut is made from the pilot hole point to the start of the tangent leader line to the start of the arc leader.

![Automatic Leader Settings](image)
The Automatic Leader Settings Dialog Box

The Automatic Leader Settings dialog box is used to define what kind of leaders you want AUTO-CODE to generate for you when the geometry is selected. Start by selecting the types of leaders you want to create. As you select the various options such as Arc, Line, or Corner, a picture will appear that will guide you in filling out the parameter and toggle options.

If no leader options are active, a blank dialog box will appear with all the optional input fields grayed out so that you cannot select anything. The only options available are a row of radio buttons that select the type of leader you want to define.

Note that the direction (CW and CCW) and side (inside or outside) can be selected in the Route Contours dialog box. This dialog box is simply to enable leaders and to define the specific parameters that define how to apply them in your drawing.

Automatic Leader Settings Dialog Box with Leader style ‘Arc’ Selected

Arc leaders support an arc that causes the cutter to enter the part at a tangent to a flat side. This type of leader is useful when cutting softer materials.

Select the leader options by changing the values in the toggles below the picture of the arc.

The Leader out option is used to enable (default) or disable a leader being added to move away from the edge of the part.

The Leader to arc option will add a linear movement in front of the arc lead in. The length of the line must be defined in the Length 2 input area next to the image of the leader arc.

The leader into the arc can come from an angle or be tangent to the start of the arc. If the Tangent to arc option is selected the line will be tangent to the entry point of the arc. This option is often used when cutter compensation is active as it provides an easy set of geometry for the tool to follow while engaging the offsets.

Selection of the Pilot hole toggle will cause AUTO-CODE to ask for the location of the pilot hole prior to starting the lead in cutting. The leader will be connected to the pilot hole so that the
cutter will move from the pilot hole to the start of the arc (or leader into the arc) and then continue along the arc to the edge of the part.

In addition to the style selections, arc leaders need to know the radius of the entry arc (entered in the **Radius/Length** field) as well as the angle (entered in the **Angle value** field). The angle is the angle of the entry (and optional exit) arc. The default angle is 90 degrees as shown in the picture. Note that the length of the leader line to the arc is defined in the “Length 2” field.

**Automatic Leader Settings Dialog Box with Leader style ‘Line’ Selected**

Line leaders are used when cutting hard materials and allow you to start a cut on a side to set up cutter compensation. The only option for the line leader is whether or not to generate a line leader away from the part at the end of the cut. Select the Leader out toggle (default) to have a line added at the end of the travel to move the cutter away from the part while turning off the cutter compensation.

Selection of the **Pilot hole** toggle will cause AUTO-CODE to ask for the location of the pilot hole prior to starting the lead in cutting. The leader will be connected to the pilot hole so that the cutter will move from the pilot hole to the start of the line leader and then continue along the line to the edge of the part.

For line leaders, supply the length of the leader line (entered in the **Radius/Length** field) and the included angle (entered in the **Angle value** field) between the entry and exit lines. An angle of 90 (as shown in the picture) results in a cut that approaches the part at a 45-degree angle.
Automatic Leader Settings Dialog Box with Leader style ‘Corner’ Selected

The corner leaders dialog box is used in a variety of machining environments and allows for a sharp corner to be cut as a result. When the corner leader is used, one or two lines are added that end at the corner point selected. The lines represent the tool movement into and away from the corner. These will be two additional moves and do not change the ending points of the shape.

Enter the length of the leader line to extend from the corner in the **Radius/Length** field. Select to have an exit leader by toggling the **Leader out** option. You can start from a pilot hole by selecting the **Pilot hole** option. As each corner is selected, the location of the pilot hole will also be requested.

**Objects supported by the leader function:**

- Closed POLYLINE and LWPOLYLINE objects
- Ellipses - converted to polyline objects when processed
- Circles - converted to polyline objects when processed
Additional Machine Control Functions

The functions found in the Machine controls section of the main dialog box will vary from one interface to another. The following commands are from the standard WIRE interface and can be used in most custom interfaces.

You can add or remove machine control functions by changing the CTL file for the interface in use. Details about the CTL file can be found in the CUSTOM.HLP file accessible from the AUTO-CODE Help menu. Machine control functions can call custom AutoLISP programs of your own design or patterns that are part of the interface in use. The functions presented below are all written in AutoLISP and can be found in the LSP file for the interface or are part of the base AUTO-CODE VLX module.

<table>
<thead>
<tr>
<th>CTL Entry</th>
<th>Function performed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dwell at point</td>
<td>Output codes to dwell at the current program point.</td>
</tr>
<tr>
<td>Comment</td>
<td>AutoLISP function for user entry of comments into the program code.</td>
</tr>
<tr>
<td>M00/M01 Pause</td>
<td>AutoLISP function to output the pause code options into the program sequence.</td>
</tr>
<tr>
<td>M98 Call Subroutine</td>
<td>AutoLISP function to call a subroutine up using the M98 code.</td>
</tr>
<tr>
<td>Cutter Compensation</td>
<td>AutoLISP function to activate the cutter compensation options of the controller.</td>
</tr>
<tr>
<td>Power Control</td>
<td>AutoLISP function to display current power settings and allow for edits.</td>
</tr>
<tr>
<td>Cut Wire</td>
<td>AutoLISP function to cut the wire.</td>
</tr>
<tr>
<td>Wire Feed in</td>
<td>AutoLISP function to thread or feed the wire after it has been cut.</td>
</tr>
<tr>
<td>Skim Cut</td>
<td>AutoLISP function to cut the same geometry multiple times to achieve a very good finish quality.</td>
</tr>
</tbody>
</table>

Dwell at current point

Outputs the dwell sequence, typically a G04 followed by an X value with the time in milliseconds to delay.

Dwell operations are typically used to wait for the power level to be achieved or for the water flow to be activated. Dwelling in the cut will result in a larger kerf at the dwell point.

Comment output

Enter text for a comment to be output in the NC program file at the current position in the program.

The comment text can exceed the window limits but should be kept to a minimum, as most machine tool controllers will only display a limited number of characters per line.
Program Operation Pause

Issue a program pause using the standard M00 or M01 syntax. Program pauses are used to allow the operator to do something while the machine tool program waits to restart after the "go" button is pressed at the controller.

Typical reasons to pause are to allow cooling to take place, to allow the operator to gauge a cut, check clamps or material for shifts, or even to load new stock manually.

Subroutine Call

Subroutines can be called up by just supplying the name. In most controller systems, the name is the program number it was stored under.

Subroutines are useful for standard operations such as loading and off-loading, locating local zero points, repositioning clamps, and starting or stopping the gas and cutting system.

Cutter Compensation Control

Cutter compensation can be controlled from the settings menu or from the machine controls selection. The method that works best for you is what to use. Both methods will result in the same operations taking place in the system.

Wire Power Control

The power control settings of the Wire interface will allow you to set the cutting power, feed rates, and activate the power. Cutting power and X-Y Feed rate for Wire cutters are often expressed in units specific to the control system.

The Electric power and Wire feed settings are provided for explicit control of the cutting tool. Some machine tools will not allow the power to turn on and off in which case those buttons in this dialog box should be ignored or will not appear in your custom interface.

Cutting power and wire feed speed are often output only in the header or start up of a Wire EDM program and will not be adjusted as the program is defined. There are always possible exceptions of course.

Cut Wire

Outputs the code(s) needed to cut the wire if available for the machine tool. Before cutting the wire, the prompt “Cut the wire <Yes>?” appears on the command line. If the Wire feed option is not active, nothing will be output as the wire is already considered cut.
Wire Feed Control

Displays a dialog box showing the current status of the wire feed and speed setting. You can also set these values in the Power Control dialog box shown previously.

Wire Skim Cut Control

Skim cutting is when the wire cutter is passed over the same geometry several times varying the offset or power setting each time. This way you can achieve a very fine surface finish.

The skim cut AutoLISP macro is for a three pass operation. Cutter compensation can be established as to the left, then to the right, then to the left again (Left-Right-Left) or the opposite (Right-Left-Right) or no cutter compensation at all (None). The offset values (Offset 1, Offset 2, Offset 3) are used at the start of each pass and are expected to include the address character (‘D’ in the screen picture) as well as the offset register value. Skim cut macros can be defined in custom interfaces using different numbers of passes and control sequences as needed by the applications.

If you know AutoLISP, you can customize these routines even more for your purposes. Customization information specific to AUTO-CODE is contained in the CUSTOM.HLP file that can be accessed from the Help menu in the AUTO-CODE pull down menu. See AutoCAD Visual LISP for help in writing AutoLISP or modifying AutoLISP programs.
Job Control Data Dialog Box

The job control data dialog box, titled Wire EDM Job Control Data, is used to collect data from the programmer about the program to be created. This information includes the job number and program number. You can also include comments that will be added to the program start. The comments are typically some note for the machine tool operator, such as the material to load or special settings that will be required.

The following NC code template shows the output that will be generated when running the standard WIRE.LSP interface. The values from the dialog box will be placed in the code instead of the heading names.

\%
Oprogram number
(job number – description)
(comment 1)
(comment 2)
(comment 3)
N10G92X0Y0
N12G90
N14G70
N16G40

Note that you can customize the interfaces so that your output may or may not be the same as what is shown above. This output shows what is obtained using the general purpose Wire EDM interface supplied with the AUTO-CODE package. AutoLISP programmers are encouraged to look at WIRE.LSP for an example of how to customize the AUTO-CODE package for your specific applications.
This page left blank