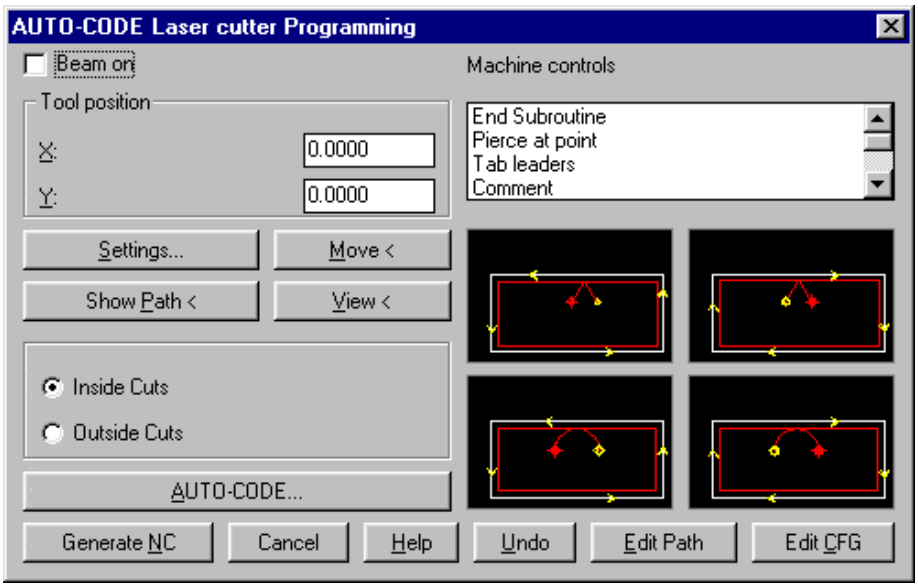
AUTO-CODE Laser Programming

This chapter explains the various components found in the standard laser-cutting interface for AUTO-CODE. Note that not every laser-cutting 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.

Laser 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 Laser cutter 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 described on the next few pages.

Beam on – Indicates if the laser or water cutting beam is currently on or off. This setting corresponds to the G0 or G1 modes of the machine tool.

Tool position – The current tool position of the cutter 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 laser variables, offsets, feed rates, and other parameters associated with setting up the machine tool can be established. When programming a laser or water jet cutter, this button is generally selected first in order to set up power levels, feed rates and offsets unless they are always used with 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 beam on toggle has been turned off.

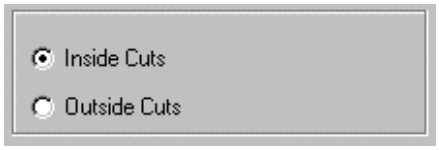
Show Path – The movement sequence of the NC/CNC path will be shown on the graphics screen. Arrows indicating the path will be drawn for each step of the program. Select a line or arc object that is part of the NC/CNC path and the sequence will continue from the selection. The following prompts will be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

```
Move to X, Y
  dX = x.xxxx dY = y.yyyy Distance=d.dddd
<Next>/Zoom/Refresh/Done/Backup:
```

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 laser 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 or technology settings as well as generating the output of the final machine codes needed to terminate a program or subroutine.

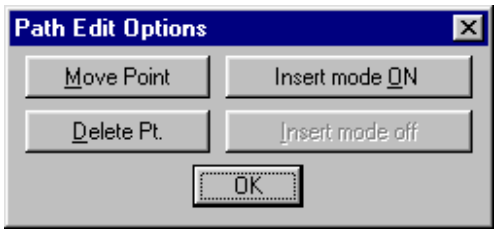
Generate NC – The Generate NC button is usually the last button clicked when writing programs inside of AUTO-CODE. A dialog box will appear asking what file name to use and then the translator will start. The translator takes the tool path information along with the settings information and creates an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

Cancel – Will exit the 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.

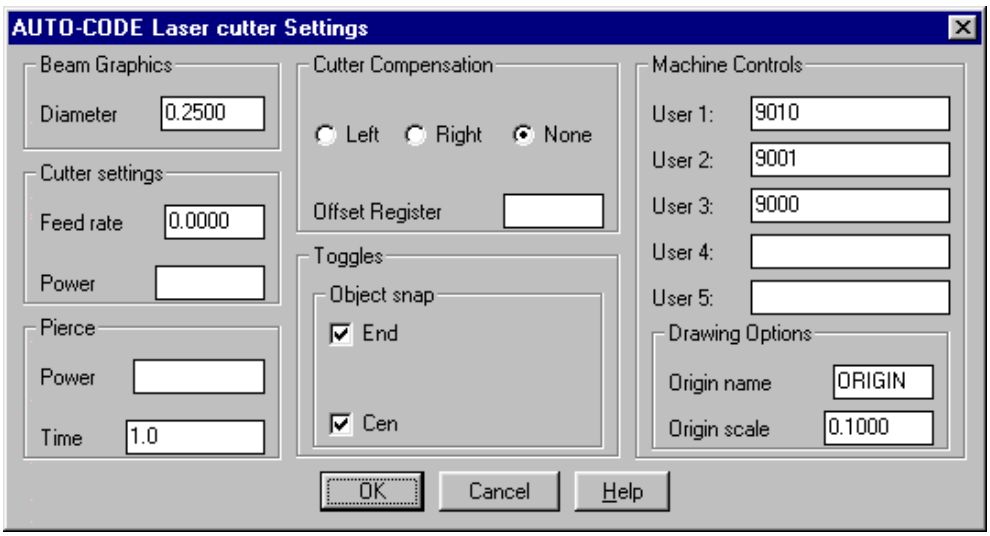
In AutoCAD operator terminology, a control point is a block containing two attributes. Do not change the structure of this block or you will cause AUTO-CODE to fail.

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 Laser cutter 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\LASER.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.

Diameter – The diameter of the cutting torch hole that is used when graphically showing the torch 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.

Feed rate – The cutting speed of the machine tool typically expressed in Inches Per Minute [IPM], Inches Per Revolution [IPR], 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.

Power – The power setting for the laser or water jet 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.

Pierce Power – The power setting to be applied for piercing the part. The machine tools interface patterns and AutoLISP utilities use this value to construct the NC/CNC output. If your machine does not use a different power setting for piercing or does not support a programmed power setting, then disregard this setting.

Pierce Time – The amount of time to delay when piercing the material with the torch. The value entered here will be used differently in each machine tool interface and in some cases will not be required at all.

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 torch 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 the 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.

Object snap – The two choices for **End** and **Cen**ter point are 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.

User 1 through User 5 – These five fields are available for customization and can take on whatever values are needed by the machine tool interface. See the CUSTOM.HLP file for more details about the user variables and how to use them. The sample interface provided uses the first and second user variables to house the numbers of the laser on and pierce and laser off subroutines (M98 call numbers).

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 box 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 track entities 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.

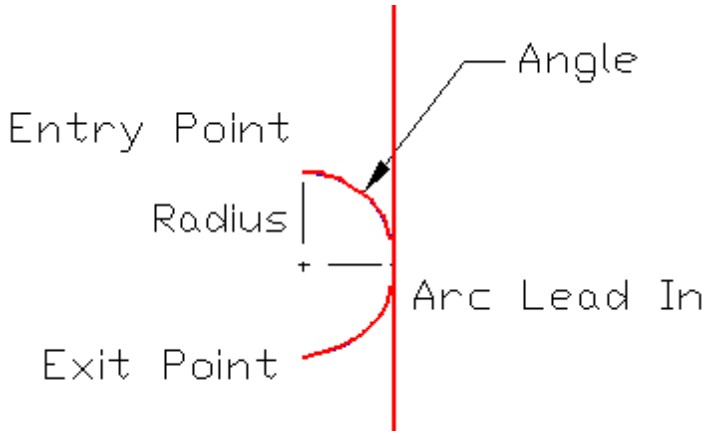
Help – Displays basic help information about the AUTO-CODE command options.

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.

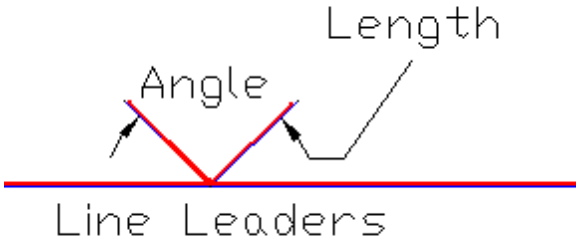


Using the Automatic Leader Feature

First select Arcs, Lines, or Corner for the style of leader. The 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.

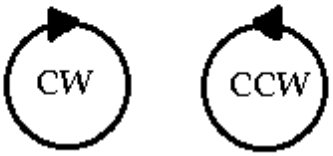
Next, select direction as either CW -clockwise or CCW - counterclockwise. This is the direction of cut around the part and is generally related to whether or not you have cutter compensation engaged and if the cut represents an outside or inside edge. The direction is also controllable from the icon 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 you may wish 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.

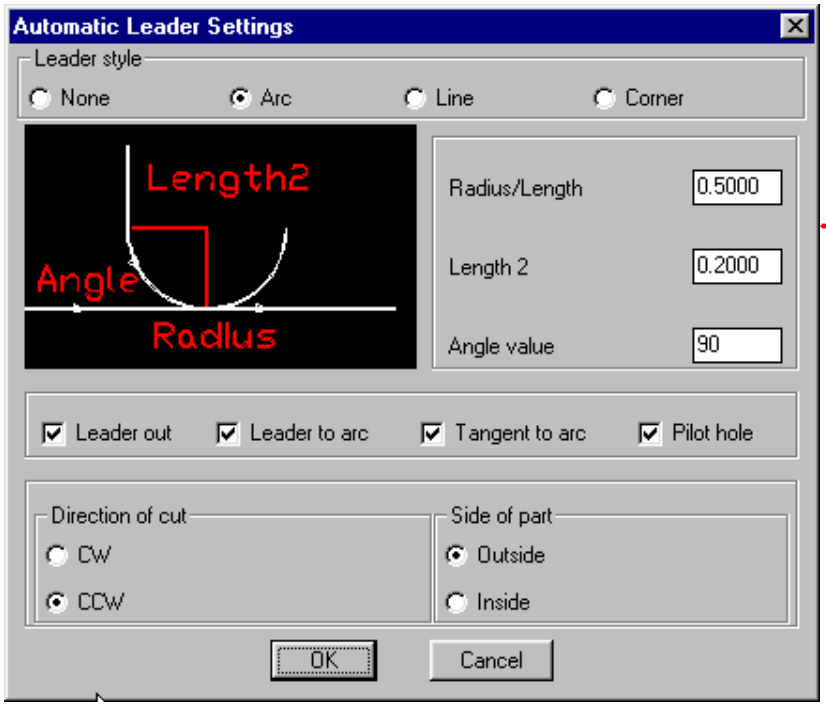


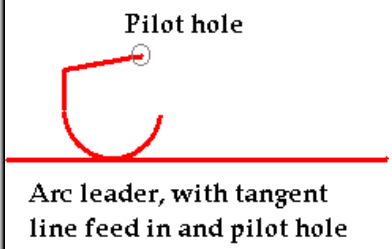
Cut Direction
The direction in which the polyline is cut is determined by a number of factors. The CW/CCW selection is only applied to closed objects that will have leaders attached. The actual direction can be changed when selecting the objects one at a time.

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 end point and start cutting.

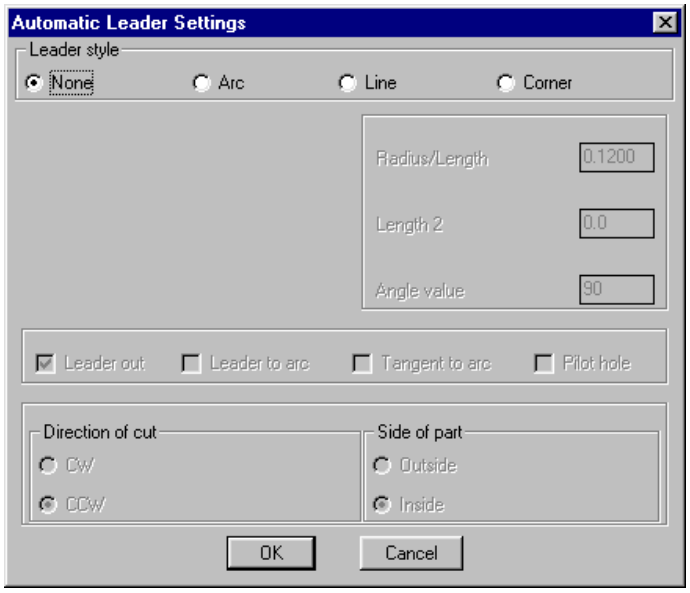
The **pilot hole** option allows you to define a starting location for the leader. A straight line cut will be made from the pilot hole location to the start of the leader.

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.





The Automatic Leader Settings Dialog Box

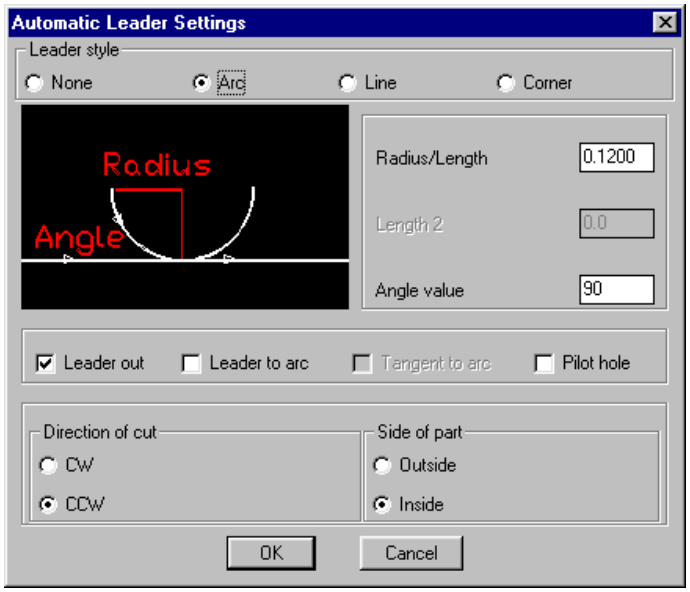


The Automatic Leader Settings dialog box is used to define what kind of leaders you want AUTO-CODE to generate for you when the geometry is selected. Start by selecting the types of leaders you want to create. As you select the various options such as Arc or Line, a picture will appear that will guide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

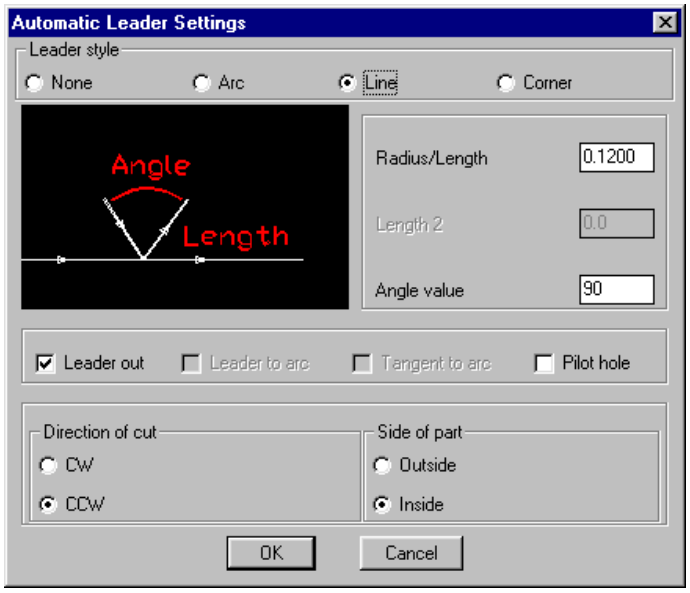
The leader into the arc can come from an angle or be tangent to the start of the arc. If the **Tangent to arc** option is selected the line will be tangent to the entry point of the arc. This

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 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

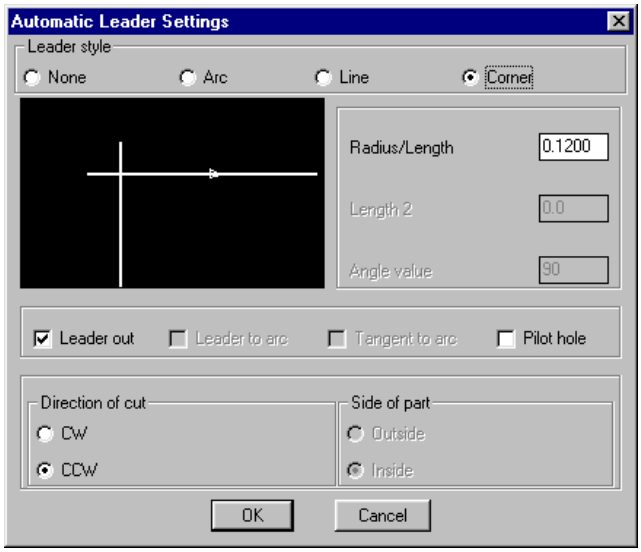


Line leaders are used when cutting hard materials and allow you to start a cut on a side to set up cutter compensation. The only option for the line leader is whether or not to generate a line leader away from the part at the end of the cut. Select the Leader out toggle (default) to have a line added 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 in the **Radius/Length** field and the included angle between the entry and exit lines in the **Angle value** field. An angle of 90 (as shown in the picture) results in a cut that approaches the part at a 45-degree angle.

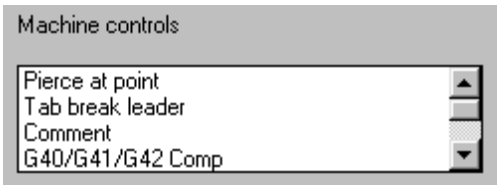
Automatic Leader Settings Dialog Box with Leader Style 'Corner' Selected



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.

Additional Machine Control Functions



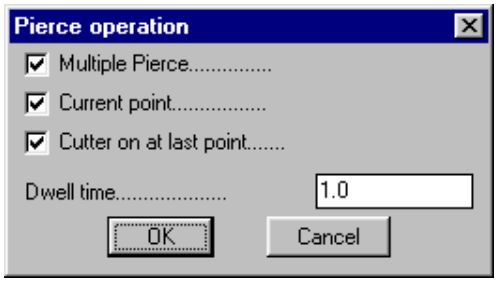
The functions found in the Machine controls section of the main dialog box will vary from one interface to another. The following commands are from the standard LASER interface and can be used in most custom interfaces.

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.

CTL Entry	**Function performed**
End Subroutine	Used only when subroutines are active, terminate a subroutine and define the next one in the sequence. See using subroutines later in this chapter.
Pierce at point	AutoLISP function to perform piercing sequence at various points.
Tab break leader	AUTO-CODE VLX function to break a polyline object and insert an arc plus line break for tab breaks.
Comment	AutoLISP function for user entry of comments into the program code.
G40/G41/G42 Comp	AutoLISP function to activate the cutter compensation options of the controller.
M00/M01 Pause	AutoLISP function to output the pause code options into the program sequence.
M98 Call Subroutine	AutoLISP function to call a subroutine up using the M98 code.

G04 Dwell	AutoLISP function to output a G04X delay in the code.

Pierce points

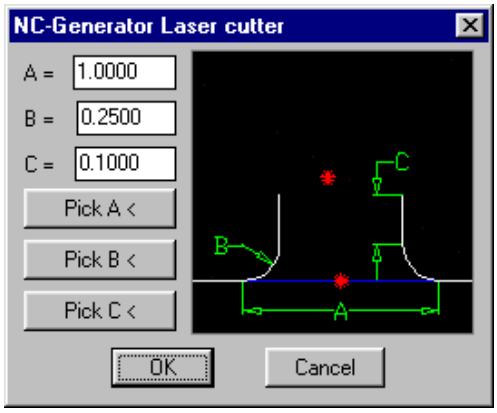


The pierce operation function allows you to select one or more points where you want to pierce the material. Select the toggles defining the desired actions. "Multiple Pierce" means you will be supplying multiple pierce points. "Current point" indicates if a pierce operation needs to be performed at the current tool location. "Cutter on at last point" is provided so that you can move from piercing directly to cutting geometry after the last point is selected. "Dwell time" is the variable delay time needed for the pierce operation to complete.

Note that you should use PILOT hole leaders when piercing is involved as a separate step and you have selected automatic leaders.

The piercing format is controlled in the AutoLISP function associated with the LASER.LSP interface and may need revised based on your requirements. The functions CNC_PIERCE_CYCLE and NCC_LASER_OFF are the specific functions that dictate the format of the operations involved.

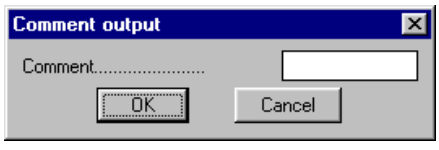
Tab Leader Option



Tab leaders are used when a cut may result in some piece becoming loose from the material too soon. The tabs will hold the cut part to the remainder of the material and can be removed in a subsequent operation or broken free after leaving the cutting table. Three parameters are supplied to the tab leader command. The distance of the gap along the original object (A), the radius of the draw out/in curve (B), and the length from the draw curve to the end of the tab (C). Enter these values or show them by selecting one of the "Pick" buttons.

After entry of the parameters, select the OK button to locate the points along the geometry to break and insert the tabs. For best results these locations should be along linear segments of the geometry and not along arc segments.

Comment output



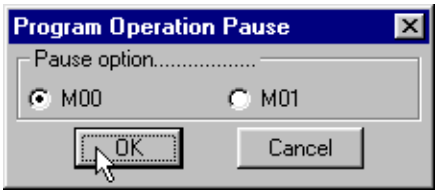
Enter text for a comment to be output in the NC program file at the current position in the program.

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.

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.

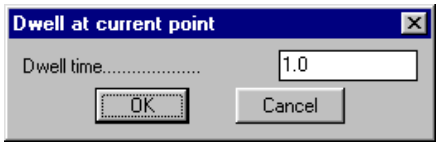
Program Operation Pause



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.

Dwell at current point

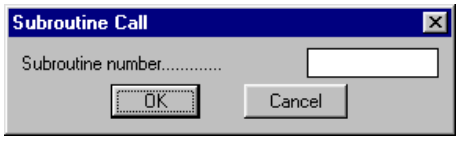


Output the dwell sequence, typically a G04 followed by an X value with the time in milliseconds to delay.

Dwell operations are used when first firing up the laser cutting device or when piercing material. They can also be used in conjunction with gas flow control.

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.

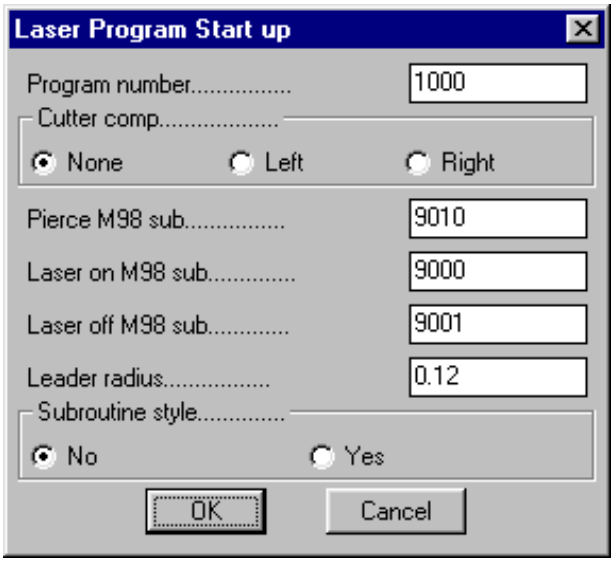


Subroutines are useful for standard operations such as loading and off-loading, locating local zero points, repositioning clamps, and starting or stopping the gas and cutting system.

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 Laser Program Start up, has two purposes. The first is to collect data from the programmer about the program to be created. This information includes the program number, cutter compensation setup and subroutine names for basic operations.



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

%
O**program number**
N1G90G54X0Y0

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 LASER interface supplied with the AUTO-CODE package. AutoLISP programmers are encouraged to look at LASER.LSP for an example of how to customize the AUTO-CODE package for your specific applications.

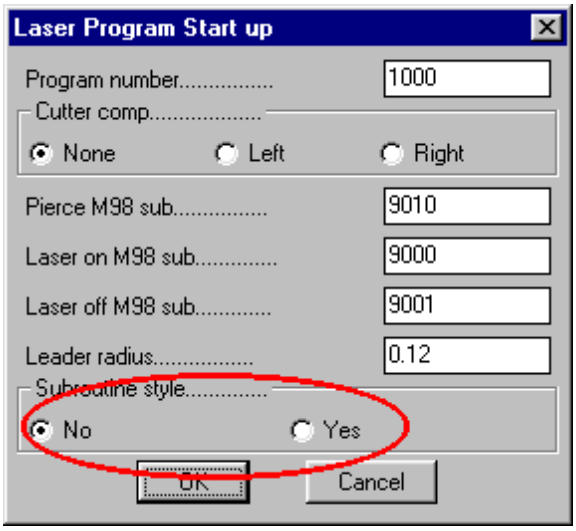
The second primary purpose of the job control dialog box is to establish the program structure you are going to generate. You can elect to use subroutines and call the subroutines numerous times or you can create a single program.

Using Subroutines

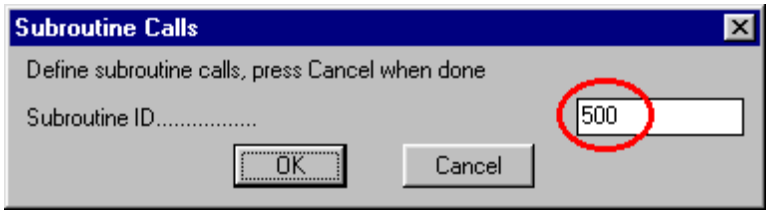
The LASER cutter interface can generate two types of program structures. The first is a standard program, with a header section followed by the cutting sequences and terminated with a standard finish section. The second is a subroutine-based format where the program starts with a header section and then proceeds to position to a point and call a subroutine. The position and call sequence is repeated as often as desired. After all position and calls have been defined, the main

program is terminated and the subroutines themselves are defined. The main program and subroutines are all contained in a single download file when the process is completed.

To set up the subroutine structure, select the "Yes" option in the Subroutine style radio option. Be sure to fill in the remainder of the header information for a proper program start up.



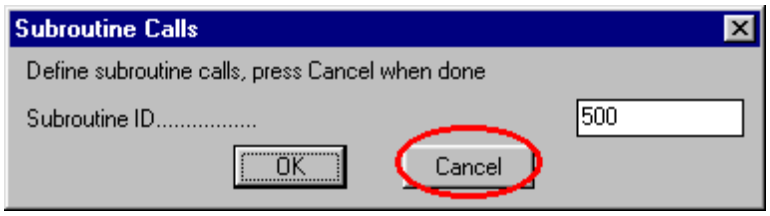
The Subroutine Calls dialog box will be displayed. The default value of 500 will be displayed as the starting subroutine number. You can change the value to anything you desire.



When you click the OK button the system will ask for a series of points that represent the call point of the subroutine you will be defining later. Just locate the start point(s) for the subroutine in the drawing that should contain all the nested figures and subroutine or insert base points.

While locating subroutine entry points press the Enter key with no point selected to return to the Subroutine Calls dialog box. The subroutine number will be incremented by one and you can define other subroutines as needed by your application.

After locating all subroutine call locations for all the subroutines you intended to define, select the Cancel button in the Subroutine Calls dialog box to move forward.



When the cancel button is selected, the main program is closed and the first subroutine to be defined is started. The starting point of the subroutine is requested. Locate the point where you will begin the definition. The point first selected as a subroutine call point is used as a default value. Locate the point or just press the Enter key to make use of the default point.

The main dialog box for the LASER interface will be shown as you begin defining your first subroutine details.

At the end of each subroutine select the "**End Subroutine**" option from the Machine Controls list. The current subroutine being defined will be terminated and the next subroutine from the list started. If there are no more subroutines to be defined you will be directed to the Generate NC button to complete the program process.