GIBBSCAM® 14

Version 14 : September 2020

Legacy Wire EDM

Proprietary Notice

This document contains proprietary information of Cambrio Acquisition, LLC ("CAMBRIO") and is to be used only pursuant to and in conjunction with the license granted to the licensee with respect to the accompanying licensed software from CAMBRIO. Except as expressly permitted in the license, no part of this document may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, electronic, magnetic, optical, chemical, manual or otherwise, without the prior expressed written permission from CAMBRIO or a duly authorized representative thereof.

It is strongly advised that users carefully review the license in order to understand the rights and obligations related to this licensed software and the accompanying documentation.

Use of the computer software and the user documentation has been provided pursuant to a CAMBRIO licensing agreement.

Copyright © 2021 CAMBRIO. All rights reserved. The Gibbs and GibbsCAM logos, GibbsCAM, Gibbs, Virtual Gibbs, and "Powerfully Simple. Simply Powerful." are either trademark (s) or registered trademark (s) of CAMBRIO in the United States and/or other countries. All other trademark(s) belong to their respective owners.

Portions of this software and related documentation are copyrighted by and are the property of Siemens Digital Industries Software.

Microsoft, Windows, and the Windows logo are trademarks, or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Contains PTC Creo GRANITE® Interoperability Kernel by PTC Inc. All PTC logos are used under license from PTC Inc., Boston, MA, USA. CAMBRIO is an Independent Software Provider.

Portions of this software © 1994-2021 Dassault Systèmes / Spatial Corp.

Portions of this software © 2001-2021 Geometric Software Solutions Co. Ltd.

Contains Autodesk® RealDWG™ kernel by Autodesk, Inc., © 1998-2021 Autodesk, Inc. All rights reserved.

DMG MORI Models provided in conjunction with GibbsCAM © 2007-2021 DMG Mori Seiki Co., Ltd.

Contains VoluMill™ and VoluTurn™ software by Celeritive Technologies, Inc. © 2007-2021 Celeritive Technologies, Inc. All rights reserved.

This Product includes software developed by the OpenSSL Project for use in the OpenSSL Toolkit (http://www.openssl.org/). This Product includes cryptographic software written by Eric Young (eay@cryptsoft.com).

Portions of this software © MachineWorks Ltd.

Portions of this software and related documentation are copyrighted by and are the property of Electronic Data Systems Corporation.

Other portions of GibbsCAM are licensed from GibbsCAM licensors, which may not be listed here.

Contents

LINKS TO ONLINE RESOURCES

INDEX

General Information

- Wire EDM Package Overview

- Wire EDM Package Detail

Wire EDM Package Overview

The Wire EDM package works in conjunction with GibbsCAM and not as a stand-alone product. It is intended to be used as a supplement to existing milling and/or turning clients. To this end, it includes functionality to produce most 2-axis parallel, 2-axis tapered and 4-axis parts. A key feature is the ease of use; therefore, some functions which are only required for special one-off situations are not directly implemented. This is to keep the interface uncluttered and to retain the ease of use. As for most things, there will normally be a work-around to handle these odd situations.

The Wire EDM package is designed to produce Wire EDM toolpath from GibbsCAM geometry. Given the fact that EDM is quite different from milling or turning, the Wire EDM package is a self-contained module that includes machining, simulation and code creation.

Using the package you will be able to create punches or dies (closed parts) and:

- 2-axis parts, with parallel walls

- 2-axis tapered parts, with or without parallel land

- 4-axis parts, with or without land

- Machine open shapes using 2-axis cuts

The 4-axis parts require two 2-axis profiles that represent the top and bottom of the tapered part of the component. Each closed part can have a user-specified number of rough/skim cuts and may have one or more gluestops.

You can machine a single part or multiple parts with Wire EDM. When more than one part is required, an overall machining strategy will dictate how the sequence of machining will occur. This will include the option to complete each part, rough each part first or rough and skim each part first.

As for the removal of gluestops, an overall strategy indicates whether they are removed after the initial rough cut, after all skim cuts, or left in the part for manual removal.

It should be noted that the Wire EDM product is a system that deals solely with the machining of wire parts and contains minimal geometry creation or editing facilities. Most geometry creation or editing must be done within GibbsCAM.

Wire EDM Package Detail

This product has been designed with ease of use as a priority. In GibbsCAM, you select the geometry that represent the parts to be cut, then load the geometry into the Wire EDM product where you can indicate how each part is machined, view the toolpath simulation and create CNC code.

To create a wire part, you must first define the geometry that represents the part boundaries within GibbsCAM using standard 2-axis geometry. Next, select all of the geometry required to define the wire part(s), including points for gluestop start locations. Wire EDM is designed to apply machining to 2D profiles; therefore, the only geometry entities it requires are 2D lines, arcs, circles and points. Once the geometry is selected, click on the *Launch EDM* option from the GibbsCAM *Wire EDM* menu to switch to the Wire EDM package.

The Wire EDM package interface is a complete window that covers the GibbsCAM window. The interface is similar to GibbsCAM but is designed for Wire EDM work.

The main functions within Wire EDM are:

- *Wire Strategy:* Defining the overall machining strategy, including:

    - Part orientation—land on top or bottom.

    - How gluestops are removed—after rough cuts, after skim cuts or manually.

    - How multiple parts are machined—complete each part, rough cuts first or rough/skim first.

    - How skim cuts are transitioned to change offsets—either by adding small lines off/on the part, or tangential arc off/on.

    - Machine configuration—travel limits, wire guide limits.

- *Machine a part*

    - Select a profile and indicate the start point of the wire for the first gluestop.

    - Indicate how many and which rough/skim cuts are to be used for the selected part.

    - Specify gluestop width.

    - Specify first element to be machined on the profile.

    - Select part type as either parallel, tapered (with optional land) or 4-axis (with optional land). For a 4-axis part, a second profile must be selected to represent the bottom of the part

- *Add a gluestop*

    - Add an additional gluestop to a machined part. Select the element for the gluestop, the start point and the gluestop width.

- *Modify the taper angle of an individual element.*

- Select any element on a 2-axis tapered part and you can change the angle of taper for that element only. Note that if the element is an arc, then the elements immediately before and after the arc must also be changed.

Using the Wire EDM product, you are able to machine either a single part or multiple parts. Each part can be either a 2-axis part with parallel walls, 2-axis tapered (with or without land) or 4-axis (again with or without land cuts).

The product enables you to allocate attributes to each part and to specify an overall strategy for all parts.

Attributes applying to each part include:

- The number of skim cuts

- The number of gluestops (and their respective locations)

- The type of part (2-axis, 2-axis tapered, 4-axis, etc.)

- The direction around the part

The overall machining strategy covers:

- When gluestops are removed—after the first roughing cut, after all the skim cuts, or not removed at all (manually removed)

- If all cuts are the same direction around each part, or alternate skim cuts are reversed (to reduce the number of times the wire need to be broken and re-threaded)

- How offsets are changed between each skim cut (by adding small moves away from the part and back on again)

- Whether all of each part is completed before moving to the next one, or if all parts are roughed and then skimmed

The strategy can be changed at any time and the toolpath will automatically be updated to reflect the changes.

Use and Reference

GibbsCAM Wire EDM is able to machine both open and closed profiles and create 2-axis or 4-axis parts.

- Each 2-axis part (parallel or tapered walls) requires a single closed profile and a point to represent the start position (wire thread location).

- Each 4-axis part requires two closed profiles, one to represent the top of the part and one to represent the bottom. You will also need a point for each gluestop start position.

- Open profiles can be machined with parallel or tapered cuts. They do not need a gluestop (it is optional) and only require a point for the wire thread position if the wire starts away from the profile.

You are also able to create No-Core Cutting. This functionality enables you to either remove a free-form shape or circular hole. Free-form shapes require a closed profile and a point for the wire start position, and holes simply require a circle.

Within GibbsCAM, you select the profiles and points required to define each part together with any profiles and/or circles for No-Core cuts. Click on Launch EDM from the Wire EDM menu and you will be automatically taken into the Wire EDM product.

> ⓘ If you do not select any geometry before clicking on the EDM option, you will be given the opportunity to start the EDM package without transferring any geometry—this is useful if you want to check/modify a previously saved Wire EDM program.

- Interface
- Wire Configuration/Strategy
- Right-Click Editing
- Post Processing Enhancements
- Creating your own posts
- Drilling

Interface

The Wire EDM interface resembles the GibbsCAM interface and uses both icons and pull-down menus. Virtually everything is done by using the icons at the top and right of the screen, but a list of the pull-down menu options is included for completeness. If you leave the mouse over an icon, you will get a short Tooltip describing the functionality of the icon. By right-clicking on an icon you will get a more detailed description.

Most of the screen area, the Workspace, is reserved for graphics. Like in GibbsCAM, the stock area is displayed with grid lines. At the bottom of the screen area are 2 boxes. Each time you select a command that requires input, the name of the command will be displayed in the first box and the type of input required will be displayed in the second box.
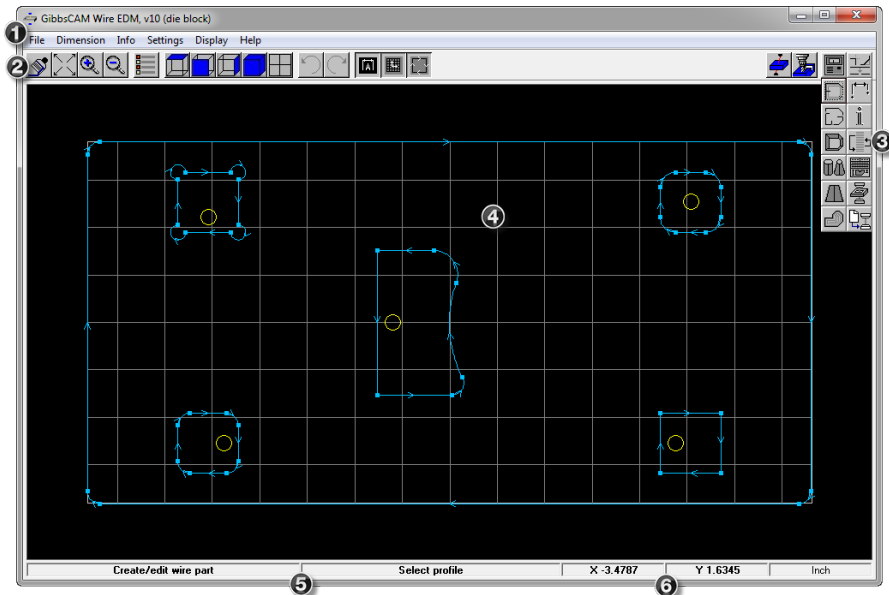
1. Menu
2. Toolbar
3. Top Level palette
4. Workspace
5. Input requirements
6. Mouse position

Elements of the Wire EDM interface.

EDM Menu Items

The pull-down menus at the top of the screen contain the following functions.

File Menu

Revert to original data
Reverts back to original geometry from GibbsCAM. All functions, commands and processes that you have executed will be erased.
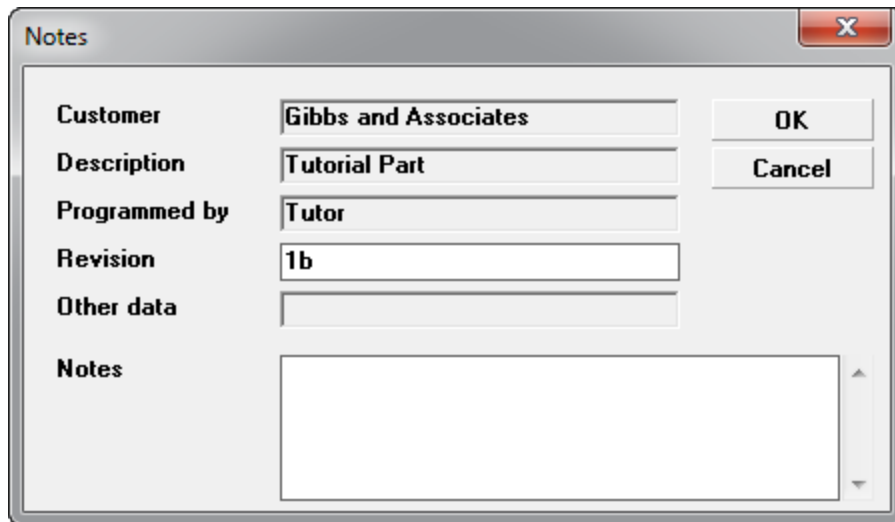
Open part
Open a previously saved Wire EDM part.

Save part
Save current part.

Save part as
Save current part as a new filename.

Notes
Select this item to enter data about the part file.

Copy (clipboard)
Copy graphics to clipboard.

Print
The Print sub-menu provides access to several print options.

Print current view
Print current view.

Print part full size
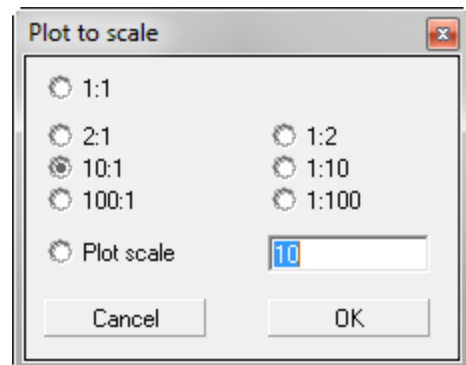Print entire full sized part.

Plot to scale
Brings up a dialog in which you can select or enter the exact scale at which you wish to plot or print.

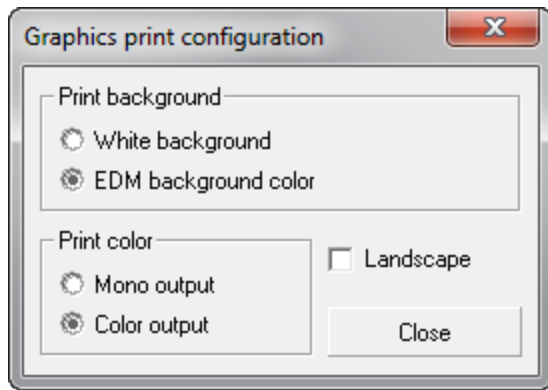### Printer set-up
Change printer settings.

Graphics print config
Configure how graphics are printed.

Exit System
Exit Wire-EDM and return to GibbsCAM.

Dimension Menu

XY Point
XY position of a point.

Line horizontal
Dimension a line using its horizontal distance (X).

Line vertical
Dimension a line using its vertical distance (Y).

Line parallel
Dimension a line using actual line length.

2 pts horizontal
Dimension between two points (horizontal distance).

2 pts vertical
Dimension between two points (vertical distance).

2 pts parallel
Dimension between two points (actual distance).

Angle of line
Dimension the angle of one line.

Angle 2 lines
Dimension angle between two lines.

Arc radius
Dimension the radius of an arc.

Arc angle
Dimension the included angle of an arc.

Configure
Configure dimension text height and decimal places.

Info Menu

Any construction element
Provides information on a selected element (point or circle).

Any non-contruction element
Provides information on a selected profile line or arc. This may include start/end point, length and angle.

Measure:
Measure the distance between two points. Select the two points and an alert dialog will open with the data requested

Area/perimeter
Calculate the area/perimeter of a profile. Select the closed shape and an alert dialog will open with the data requested.

Calculator
Brings up the desktop calculator that includes memory/recall and trig functions.

Settings Menu

Points
Show/hide gluestop start points/circles.

Dimensions
Show/hide dimensions.

Axes
Show/hide axes (XY, XZ or YZ) symbol.

Language
Select operating language.

Interface
Select the interface style you prefer to work with. Gibbs style - fixed is the default and locks the Top Level palette in the top right corner. Gibbs style - floating unlocks the Top Level palette so it may be placed anywhere on your screen. The items in the Icons menu let you enable and disable controls found on the EDM Toolbar.
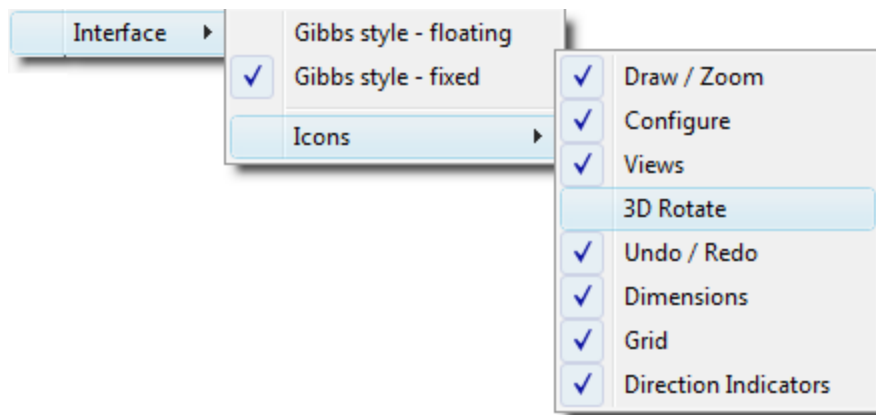
Setup post editor
Allows the user to assign specific preferences for post editing.

Setup technology data
This dialog allows you to define a list of material types, wire types and wire diameters. In Wire strategy/configuration settings, drop-down menus are used to select material and wire. The corresponding technology data entered here will be automatically loaded into those lists.

Hot keys
Customizes hot key commands.

Display

Redraw
Redraw will force a screen refresh that will clean up the display of items.

Full
Zoom full will shrink or grow the display of items in the Workspace to fit within the Wire EDM window.

Window
Select this item for a drag-zoom.

Zoom in
Zoom in by 10%.

Zoom out
Zoom out by 10%.

Pan left
Pan left by 10%.

Pan right
Pan right by 10%.

Pan up
Pan up by 10%.

Pan down
Pan down by 10%.

Viewports

Select the multiple viewports configuration. The Viewports command can split the screen into two, three, or four views. The setting is controlled by the Select Viewport layout dialog which can also be accessed from the Multiple Viewports button. Simply click on one of the six configurations to change the Workspace display. Each viewport can be individually set to the view you prefer.
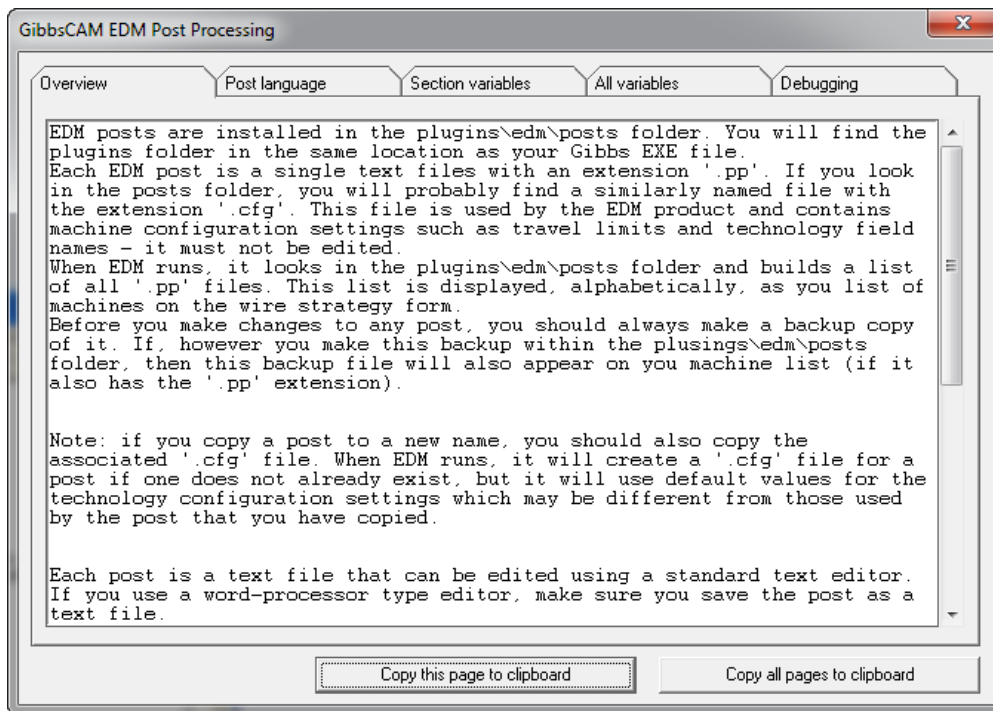
Help

About GibbsCAM Wire EDM

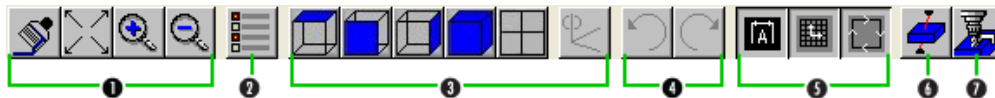Displays information about the product and the version currently being used.

Posts

Displays a help window (shown below) with topics relating to EDM post processing.

EDM Toolbar

The Toolbar provides quick access to commonly used commands such as setting the display and view settings, part configuration and undo/redo functions.

1. Display commands
2. Configuration
3. View commands
5. Miscellaneous commands
6. Solid Rendering
7. Return to GibbsCAM

4. Undo / Redo

Display Commands

The first four buttons control the display of items in the Workspace, similar to several of the items in the GibbsCAM View palette. These commands include Redraw, Zoom Full, Zoom In and Zoom Out.

Redraw
Redraw will force a screen refresh that will clean up the display of items.

Zoom Full
Zoom Full will shrink or grow the display of items in the Workspace to fit within the Wire EDM window.

Zoom In
The Zoom In command will increase the display of items in the Workspace by 10%. You may also Zoom In on the part by using `Shift + "+"` or the mouse wheel or by dragging the mouse on an area that you want to zoom in on.

Zoom Out
The Zoom Out command will decrease the display of items in the Workspace by 10%. You may also Zoom Out on the part by using `Shift - "-"` or the mouse wheel.

In addition, `Ctrl + arrow keys` will pan the part in the specified direction. Right-clicking a part and dragging it will also move it in the specified direction.

Configuration

The Configuration button opens the Configuration dialog which provides for updating configuration settings, including display settings on elements.
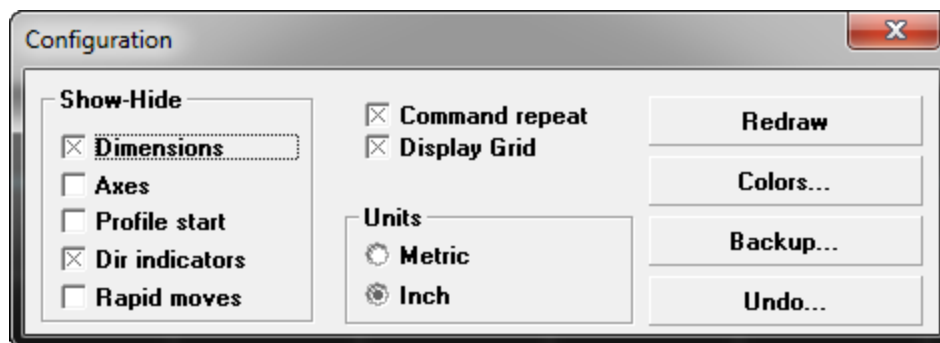
Show-Hide
This set of checkboxes controls what is displayed in the Workspace. Checking Dimensions will display the dimensions created using the Dimension icons, while Axes display the part's axes as in GibbsCAM. Profile start will draw an arrow indicating the direction in which a cut will start. Dir indicators is a series of arrows showing the direction of a cut, drawn along geometry and toolpath. Checking Rapid moves will display rapid moves as dotted lines so you can see where the machine is moving.

Command Repeat

Activating the Command repeat checkbox allows you to perform a function, such as breaking geometry, multiple times without having to re-select the command. When a function, such as Split Element, is chosen, the system will remain in that mode until another function is selected. This is very convenient when performing a series of commands using the same function.

Display Grid

Activating the Display Grid checkbox draws the stock boundaries using grid lines.

Units

These radio buttons toggle between Metric or Inch measurement units.

Redraw

After making the above selections, click Redraw to instantly see the results of your changes without closing the Configuration dialog.

Colors

This dialog allows you to configure the display colors of the Wire EDM system. Click on the name of the item to change the color. Check Use wire color for each wire cut to be rendered in a different color (if unchecked, all machining is rendered in the same color).

Backup

This setting allows you to designate the backup file and frequency of backups. Select the Autobackup enabled checkbox to activate this function.

Undo

This setting allows you to set the number and types of undos the system tracks. Each command is classed as a type 1 or type 2 command. The commands that have a greater effect are classed as type 1 (like creating a machined part), those that have a lesser effect are type 2 (like fillet). For large jobs, using Undo can slow the system down, so there is the option of just storing undo information for the more serious commands (level 1) which will speed things up a little. the number of undos affects the amount of disc space used to store undo data. It is recommended to leave the number of undoes at 100 and setting the system to Undo level 1 and 2 commands. If you are very low on disc space, you could reduce the number of undos and if your system is very slow, you could try undo level 1 only or even no undo.

View Commands

The View control buttons set the view orientation of the part geometry. Settings include Plan, Front, Side, Isometric, Multiple Viewports, and Mouse View Rotation.

Plan

The Plan button sets the view to the top of the part, looking down on the XY plane.

Front

The Front button sets the view to the front of the part, looking at the XZ plane.

Side

The Side button sets the view to the top of the part, looking at the YZ plane.

Isometric view

The Isometric button sets the view to a three-axis view of the part. While in isometric view, click the Mouse View Rotation button and the cursor will change into an axes icon when it is positioned over the part in isometric view. Left click and hold the mouse scroll wheel to dynamically rotate the view. Remember to deactivate this feature after you have finished with it by right clicking or by clicking on the icon again. Additionally the view may be rotated without the Mouse View Rotation button by using the mouse scroll wheel.

Multiple Viewports

The Multiple Viewports button can split the screen into two, three, or four views. The setting is controlled by the Select viewport layout dialog which is accessed from the Multiple Viewports button. Simply click on one of the six configurations to change the Workspace display. Each viewport can be individually set to the view you prefer.
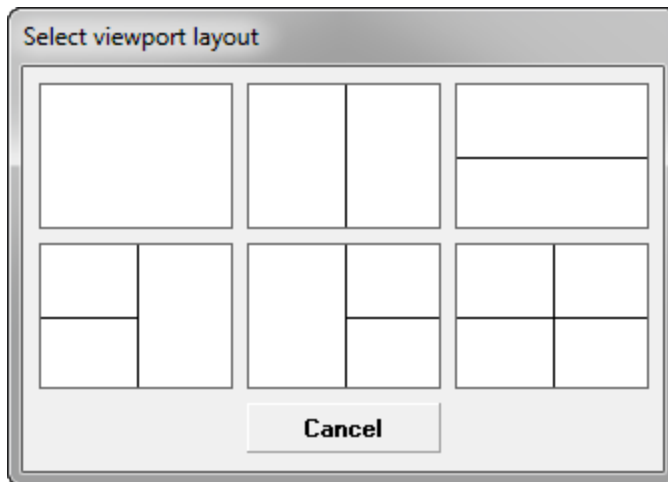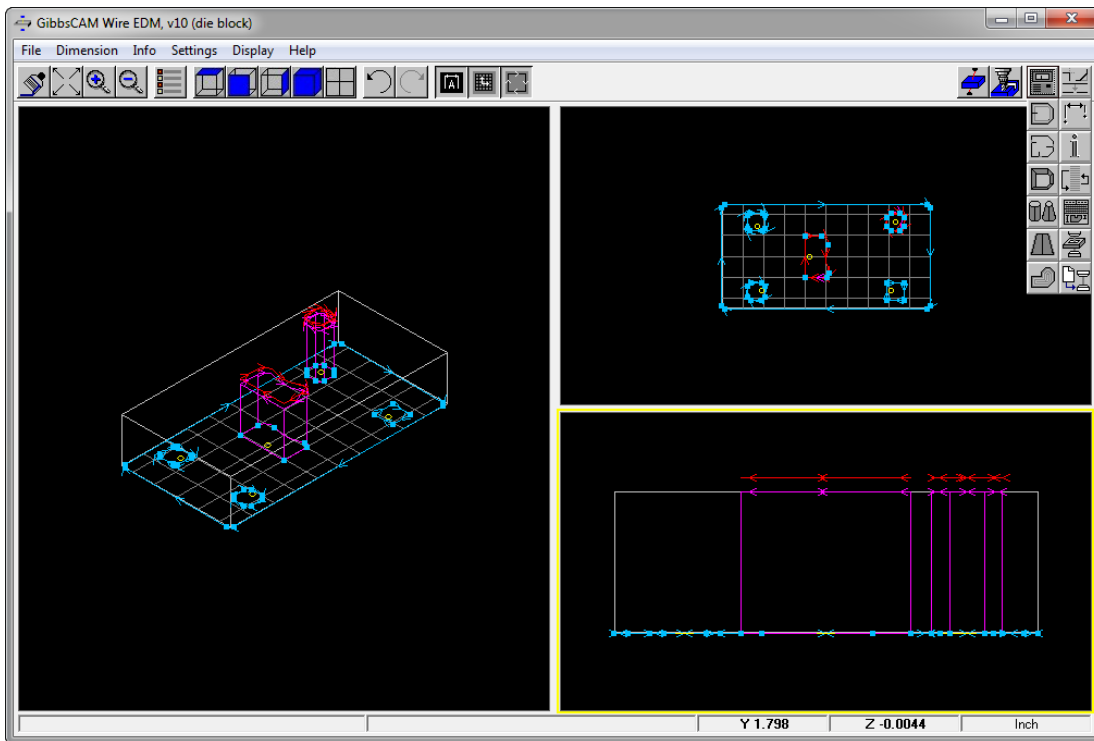
The Select viewport layout dialog and a part displayed with multiple viewports

Dynamic View Rotate

This item is used with Multiple Viewports. When you left-click in one of the windows the part will rotate with the mouse. Right-click to stop the rotation.

Undo/Redo

The Undo and Redo buttons allow you to remove changes on a part. The undone changes can also be re-done. The Undo command logs the last 100 modifications made to the part file including syncs, geometry changes and machining functions. The Undo function may be fine-tuned or disabled in the Configuration dialog.

Solid Rendering

Clicking on the Solid Rendering button will change the stock diagram to a 3-dimensional solid image. When you click the Solid Rendering button new icons will appear in the Top Level Palette. These buttons provide control over Solid Rendering. The view can be rotated, panned and zoomed as the part is being cut. Clicking Play will begin the Solid Rendering. Clicking on Step Forward moves the rendering forward in small increments. The remaining buttons are Stop, Rewind and color adjusting slides.
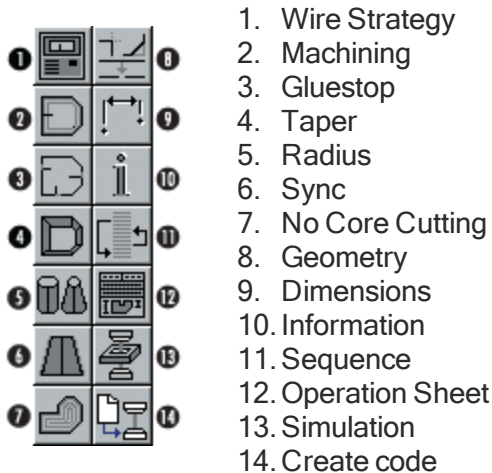
Return to GibbsCAM

Clicking the Return to GibbsCAM button will switch the user back to the GibbsCAM environment where the user may make changes to the geometry. Selecting the new geometry and re-launching EDM will send the user back to the Wire EDM environment,

Top Level EDM Palette

This palette contains icons to access setting the wire strategy and machining settings.

1. Wire Strategy
2. Machining
3. Gluestop
4. Taper
5. Radius
6. Sync
7. No Core Cutting
8. Geometry
9. Dimensions
10. Information
11. Sequence
12. Operation Sheet
13. Simulation
14. Create code

Wire Strategy/Configuration

Clicking on this button opens the Wire strategy/configuration settings dialog. This dialog allows you to set the machining methodology, wire guide locations, 4-axis display settings and any parameters that relate to the whole job. The Wire strategy/configuration settings dialog is discussed in greater detail under Wire Configuration - Machine Configuration Tab.

Machining

Clicking on this button allows you to create or edit a wire part. Select the geometry that represents the profile to be cut. On 4-axis parts, the top profile should be selected. More information may be found in Creating a Wire EDM Part.

Add or Edit Gluestops

Clicking on this button allows you to add, edit or delete a gluestop. Select an element (line or arc) on an existing wire part where you require an additional gluestop to be created. Select the start point for the wire and a gluestop will be added. If you select an element that represents an existing gluestop, you will be able to change the gluestop attributes (gluestop width) or to remove the gluestop altogether. More information may be found in Add or Edit a GlueStop.

Modify Taper

Clicking on this button enables you to modify the taper angle of individual or a range of elements on a 2-axis tapered part. Simply click on the element of the part that you want to change and you will be shown the overall part taper angle and the angle for the selected element. Enter a new angle and you will see the part change accordingly. More information may be found in Modify Taper Angle.

Specify Conic or Cylindrical Radius

Clicking on this button enables you to specify whether the radius of an arc is conical or cylindrical or you may enter your own radius value. Cylindrical arcs have a constant radius while conic arcs have a larger radius at the end of a taper. This option can be applied to tapered parts only.

Sync

Each 4-axis part is automatically synchronized to end points of an element on the selected top and bottom profile(s). This icon enables you to add additional sync points around the profile. Click near the end of an element on the top profile, then as you drag the mouse around you will see a line being rubber-banded from the end of the element. Click near the end of the element that you want to sync with on the lower profile and the machining for that part will be re-synchronized. More information may be found in Wire Synchronization.

No-Core Cutting

Once you have clicked this icon, you select either a closed profile or a circle. If you select a profile, you will be asked where you want to start (and must select a point), the wire overlap and the number of skim cuts as well as selected skim cuts. If you select a circle, you will be asked for the initial hole diameter and the hole will be machined by a series of ever-increasing arcs to remove the material between the initial diameter and the final (circle) diameter. More information may be found in No-Core Cutting.

Edit or Modify Geometry

Clicking on this button enables you to edit or modify existing geometry.

Commands include Fillet, Chamfer and Split an Element. More information may be found in Dimensioning the Part.

Dimensions

Clicking on this button will activate the Dimension dialog. This will allow you to add dimensions to your part.

Options include Line Horizontal, Line Vertical, Line Parallel, 2 Points Horizontal, 2 Points Vertical, 2 Points Parallel, XY Point, Angle of Line, Angle of 2 Lines, Arc Radius, Arc Angle and Text Configuration. These items are also accessed from the Dimension menu. The descriptions of each item are under Dimension Menu and more details on dimensioning a part may be found in Dimensioning the Part.

Information

Clicking on this button will activate the Info dialog.

The Info dialog has several utilities. Commands include Any Construction Element, Any Non-construction Element, Measure, Area/Perimeter and Calculator. Select an element to get data relating to its start point, end point, angle etc. More information may be found in Information.

Resequence the Operations

Resequence the order of machining. The machining is normally carried out in the order in which you have created it. To change this sequence, click on this icon and simply select each part or No-Core shape/circle that you want to machine in the order that you want to cut them. If you only select some parts, then those selected will be machined before the others that were not selected.

Note that all No-Core cuts will always be performed before the parts are cut. More information may be found in Resequence the Operations.

Operation Sheet

Clicking on this button generates an Operation sheet that includes part notes, a listing of all operations and a part preview. The content may be customized by selecting the Notes item from the File menu. The data entered in Notes will be reflected in the Operations sheet.
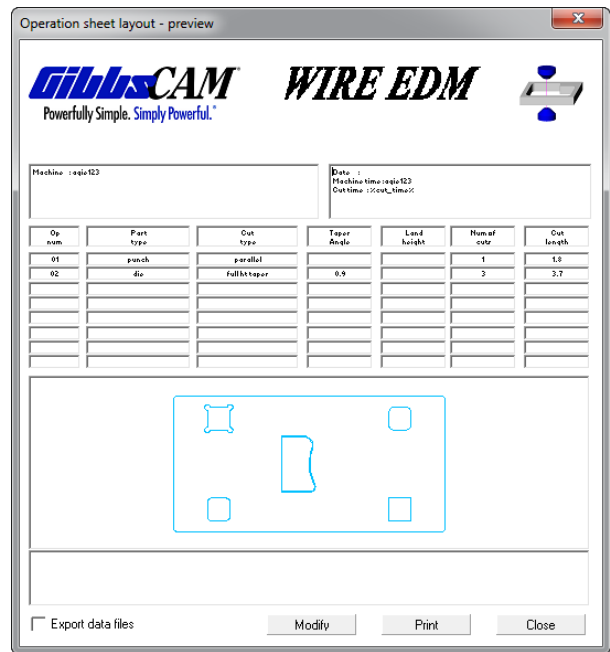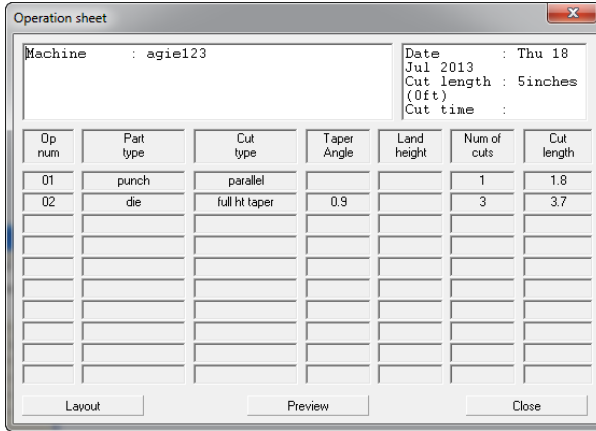
Machining Simulation

Click this button to simulate the machining. You can select the speed of simulation using the scroll bar at the top of the dialog. You can also switch to Step mode, so that you can check each individual movement of the wire. More information may be found in Machining Simulation
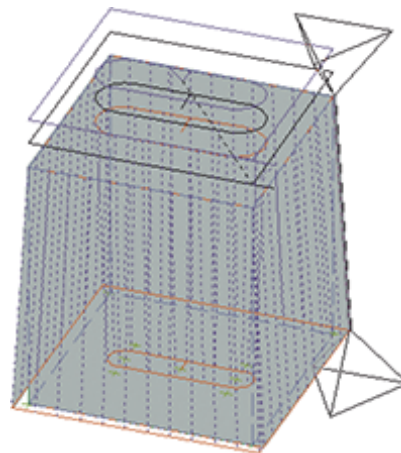

The Simulation dialog and a rendered part

Create CNC code

Clicking this button will allow you to simultaneously generate the CNC code and view the Cut Part Rendering. When Go is clicked, you will be prompted to enter a program number. When you click OK after entering a program number, the rendering and code generation will begin. The

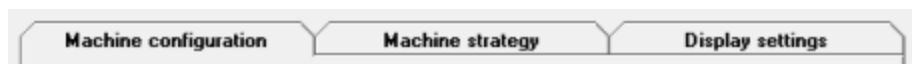code is saved to the file "W1" on your C:\ drive by default. More information may be found in Creating CNC Code.
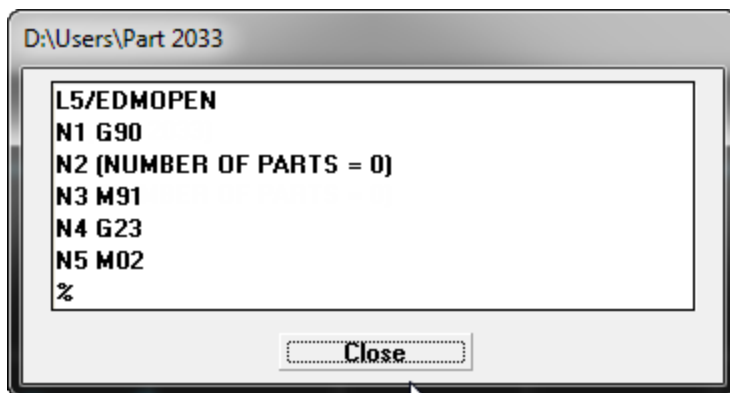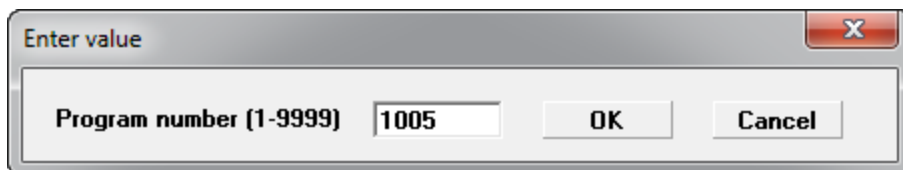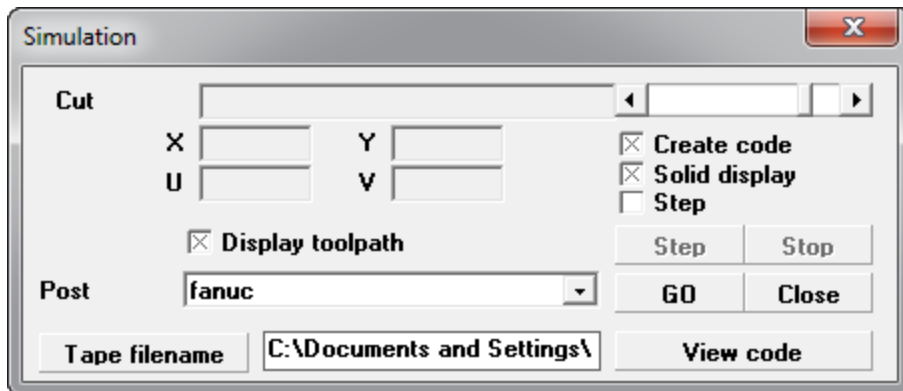

Dialogs involved in creating code

Wire Configuration/Strategy

The Wire Strategy/Configuration Settings dialog is split into three areas, each selected by clicking on the relevant tab near the top of the dialog.

Wire Configuration - Machine Configuration Tab

The settings found in the Machine configuration tab enable you to set the parameters for the machine that will be used to cut the part.
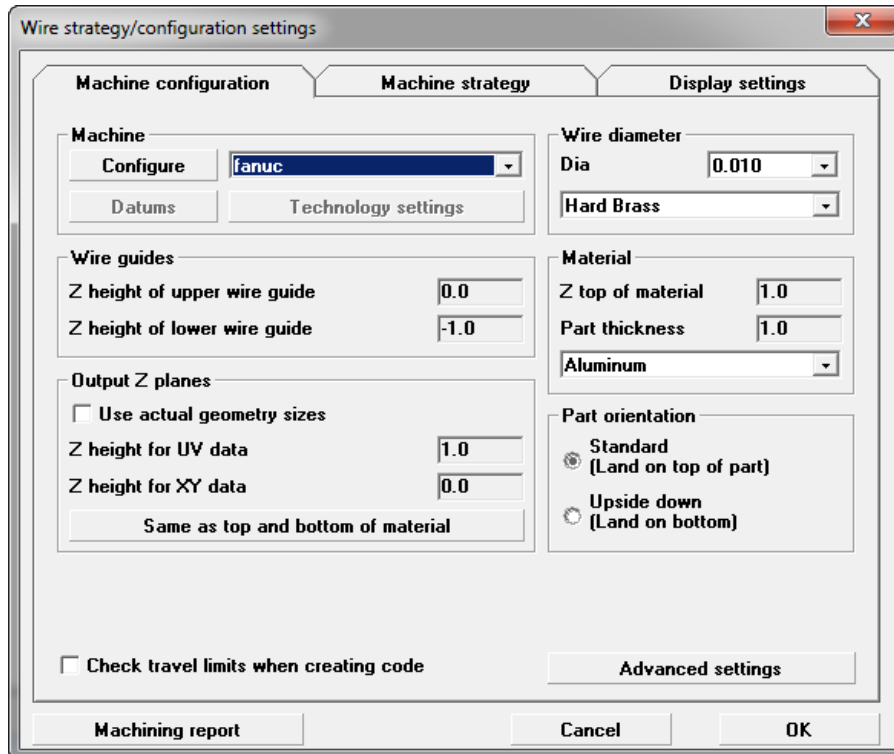

Machine

Select the machine that you will be creating code for. A list of machines configured on your system will be available on the pull-down menu. If you click on the Configure button to the left of this menu, you will be able to modify parameters specific to the selected machine, including machine travel limits and wire guide limits.
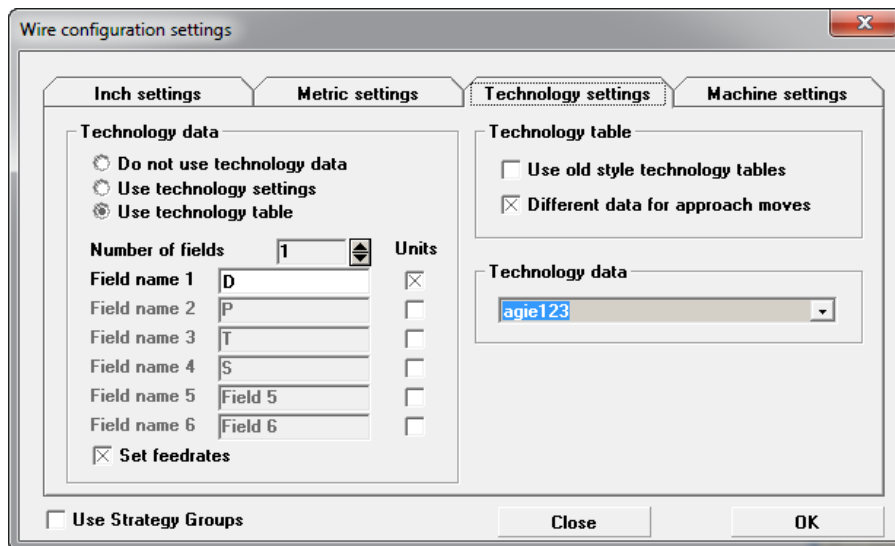
Configure
Pressing this button opens the Wire configuration settings dialog that allows you to set the physical travel limits of the wire guides and the wire's taper limits.
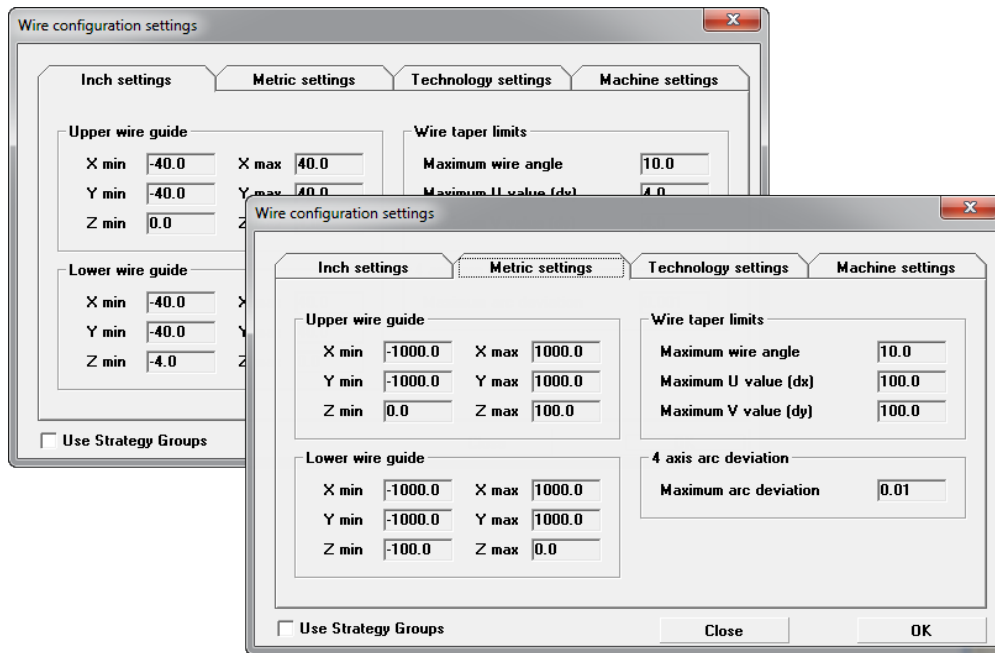
Technology Settings

There are three options in the Technology data under the Technology settings tab: Do not use technology data, Use technology settings, and Use technology table.

For the latter two options, you can specify up to six different technology values for each machine you have. Select the number of values you require by using the scroll arrows next to Number of fields. Clicking on the box to the right of the Field name allows you to enter your own field name. For example, you may wish to enter Offset diameter, Feedrate, Flushing or Epak number. Once you have configured how many technology fields you want to use together with their corresponding names, you can then enter the actual data to be used. If you check the units box, EDM knows that particular field is an inch/metric value and that you will be defining the content using the current units (inch or mm). If you use that table later in a part that has the opposite units, it will either multiply or divide the value by 25.4 to give the required data.

When employing the Use technology settings option, this is done by clicking on the Technology settings button within the Machine section of the Wire strategy/configuration settings dialog.

The above table enables you to provide values for each technology field against each wire number that you will use to machine the parts. You can specify up to eight different wire numbers, one roughing and seven skim. The values entered in this table will be available next time you start the system; alternatively, you can save specific sets of data and reload them later and therefore build up a library of technology data for different material types, thicknesses, etc.

When Use technology table is selected, click the Technology table button within the Machine section of the Wire strategy/configuration settings dialog to define the technology table using the dialog shown above.

A technology table contains a number of pre-defined machining methods which indicate how many skim cuts are to be used and the settings associated with the rough and each individual skim cut. In the example shown above, we have selected Rough + 1 Skim and entered data for these two cuts. Once the technology table is defined, click Save table to save it for future use.

Set Feedrates
Set Feedrates checkbox at the bottom of the Technology Settings dialog enables the input of feedrates against each wire offset. If this checkbox is enabled, the Machining Report button at the bottom of the Wire strategy/configuration settings dialog becomes active. The Machining Report is also discussed in Machining Report.

By using this command, you can quickly see how much time elapses until the first gluestop. You can simply select a new strategy and get a new estimate to judge which might be the best strategy under the circumstances.

Machine Settings
The Machine settings tab is used to configure datum settings.
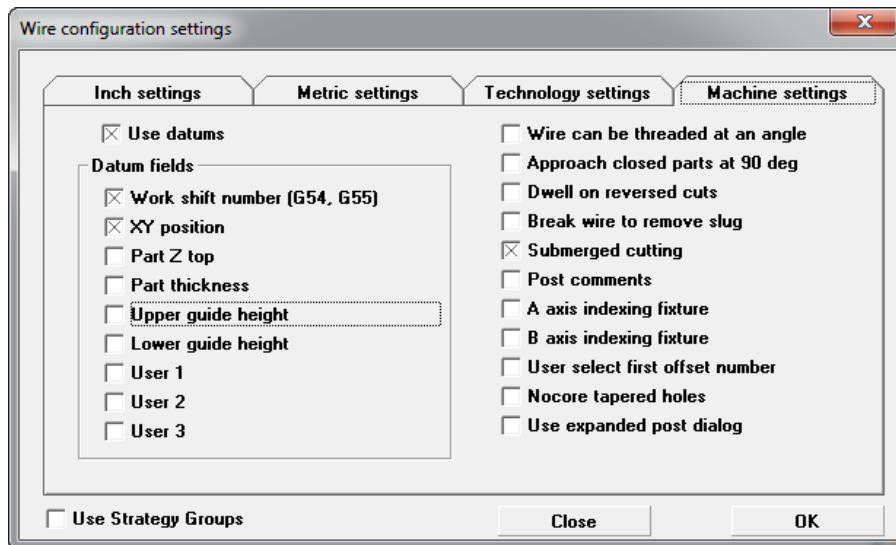
Select Use Datums to define up to ten sets of datum. The Datum fields section can be used to select the desired fields by selecting the corresponding checkboxes. When datums are enabled, the Datums button becomes available under the Machine section of the Wire strategy/configuration settings dialog. Click this button to access the Wire Datum Settings dialog as shown here.
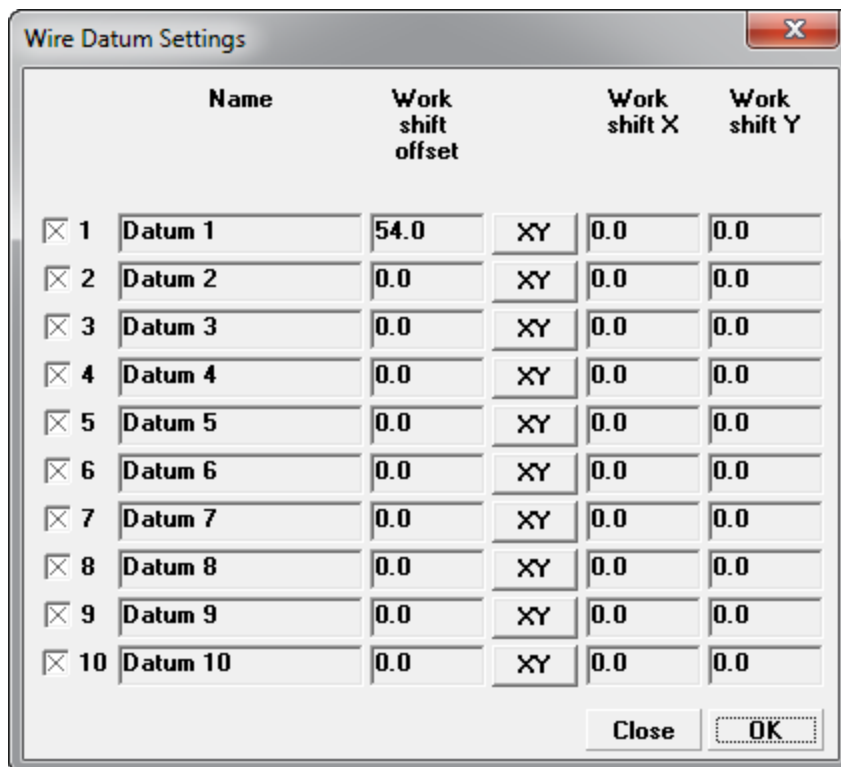
You simply check the datums you want to use. Allocate a unique name to each datum, and then enter the data associated with the datum. If you checked the XY position field, you will be offered an XY button–click on this to select a point from the screen to automatically fill in the Work shift X and Work shift Y position boxes.

After defining each datum, you will see a new right-click menu the next time you right click on a machined shape to change its properties. The right-click menu contains a list of the names of the datums you have defined. Select the required name in the list to associate the shape with the required datum.

If you are using the Z top and Z thickness datum attributes, the part will be displayed at its new Z position the next time you redraw—this provides the ability to program stepped plates with parts of different thicknesses (you can also define optional new wire guide heights).

All of the datum parameters you define are available to the post processor, including the ability to automatically shift XY positions by the defined values, if required.

Wire can be threaded at an angle:
When Wire can be threaded at an angle is checked, the wire can be threaded when the two wire guides are not directly above one another. Wire start positions are defined by XY and UV positions. When you define a 4-axis part with this option selected, the system will ask you for a start position on both the top and bottom profiles.
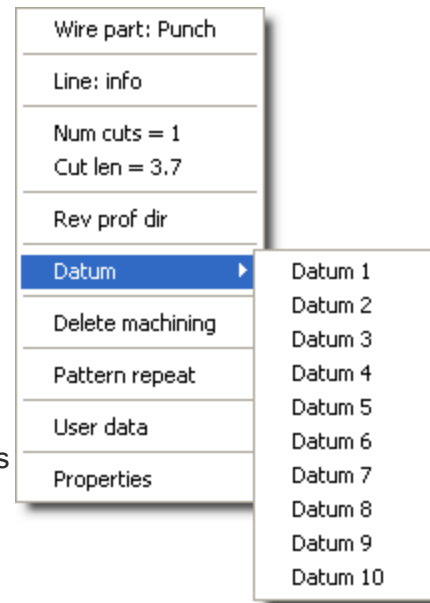
Wire Diameter

The Dia is the diameter of the wire you will be using. The different wire types that may be used are available on a pull down menu.

Wire Guides

The Z height of the upper and lower wire guides are used both for simulation of the machining and also to calculate the XY/UV coordinates for 4-axis code.

Material

The thickness of the stock and the Z position of the top of the stock should be entered in these text boxes. This is important for both drawing geometry and generating output data. The different materials that may be used are available on a pull down menu.

Output Z Planes

These entries are used when generating code. It is recommended that you click on the Same as top and bottom of material button to automatically set these values based on the material settings.

The system outputs code based on the Z height of XY data. If your part is on a block, a custom arrangement or below the top of the table, you will need to enter the proper values manually.
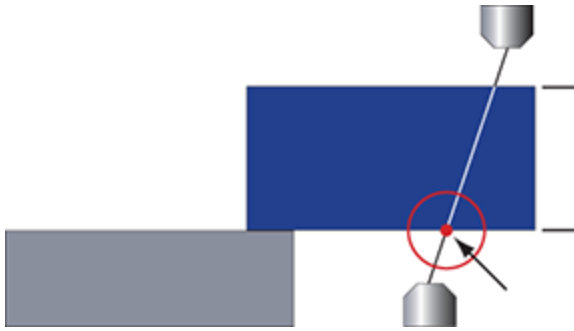
Output is done at the Z height of the XY data

Part Orientation

Specify how the part is situated. Rendering will be modified to match your setup.

Check Travel Limits When Creating Code

If this option is checked, you will get a warning message if you try to move outside the travel limits you set for your machine.

Advanced Settings

The Advance Settings button opens a dialog that contains additional strategy options for control over three areas of machining, No-Core circles, Circular dies and the entry moves made on Rough cuts.

No-Core Circles - Skim cut entry positions
This option is similar to that for circular dies (below). The skim cut entry positions may be adjusted for No-Core operations on a circle.

> (i) To get skim cut entry position evenly spaced, the following conditions must be met:
>
> - You must select the machining strategy to remove gluestops with the rough cut.
>
> - The shape must be a circle (not a circular contour).

Rough cut entry moves
This option offers the facility to select the type of entry move for the rough cut for each shape. The line and arc option will require the input of an arc radius (which will be used for all entry moves).

When a line and arc entry is selected, the line will start at the wire thread position and end tangent to the arc (to form a teardrop shape). If the wire start position is less than the arc radius from the shape, then the line cannot be tangential to the arc (as it will start inside the arc) - in such cases 90 degrees of arc will always be used.

For parts with land, a line-arc will only be used for roughing the land, unless the skim taper options has been selected on the second page of wire strategy settings.

> There is no gouge checking for any arc entry moves, the user must ensure that gluestops are not placed such that the arc will cut through any other part of the shape. In cases where land parts have line-arc entries for both land and taper, the user must check that the two arc entries do not create any loose material which may fall from the part.

Circular dies - Skim cut entry positions:

When gluestops are removed with the roughing cut, dies will have the slug removed before any skim cuts. For circular dies, we have added the option to adjust the entry moves for each of these skim cuts such that each skim will start at a different position. The entries positions will be equally divided around the circular shape, so for 3 skim cuts they will be 120 degrees apart, 4 skim cuts 90 degrees apart and so on.

Each skim cut will comprise 2 circular arcs (of equal length) and each skim cut will go past the previous exit position. So for 3 skim cuts, each skim cut will comprise two 240 degree arcs (to cover 360 degrees, plus 120 degrees to the next skim cut start position).
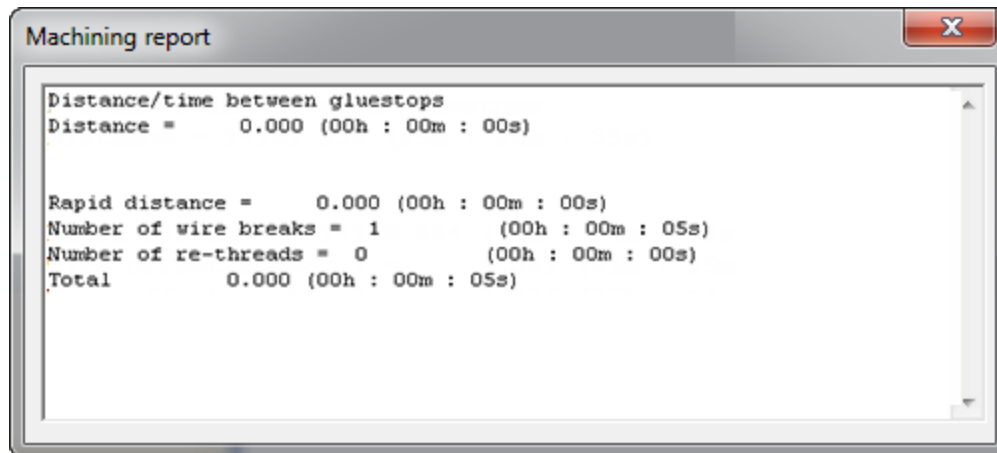
> To get skim cut entry position evenly spaced around a contour, the following condition must be met.
>
> - You must select the machining strategy to remove gluestops with the rough cut.
> - The shape must be closed circular contour.
> - The shape must be a die (start point inside the circle).

Machining Report

The Machining Report command quickly calculates how much time elapses until the first gluestop. This option uses the feedrates together with the selected strategy to provide a simple report showing the total estimated time, time per wire offset, and distance cut with each wire offset. It will also list the time between each gluestop.

```
Machining report                                    X

Distance/time between gluestops
Distance =      0.000 (00h : 00m : 00s)


Rapid distance =      0.000 (00h : 00m : 00s)
Number of wire breaks =  1          (00h : 00m : 05s)
Number of re-threads =  0          (00h : 00m : 00s)
Total          0.000 (00h : 00m : 05s)
```

Wire Configuration - Machining Strategy Tab

This is where you define the strategy to be used for the whole job. Click on the second tab, Machine strategy, you will get the following options:

Machine Strategy

The settings found in the Machine stategy tab enable you to set the parameters for the cutting of the part.
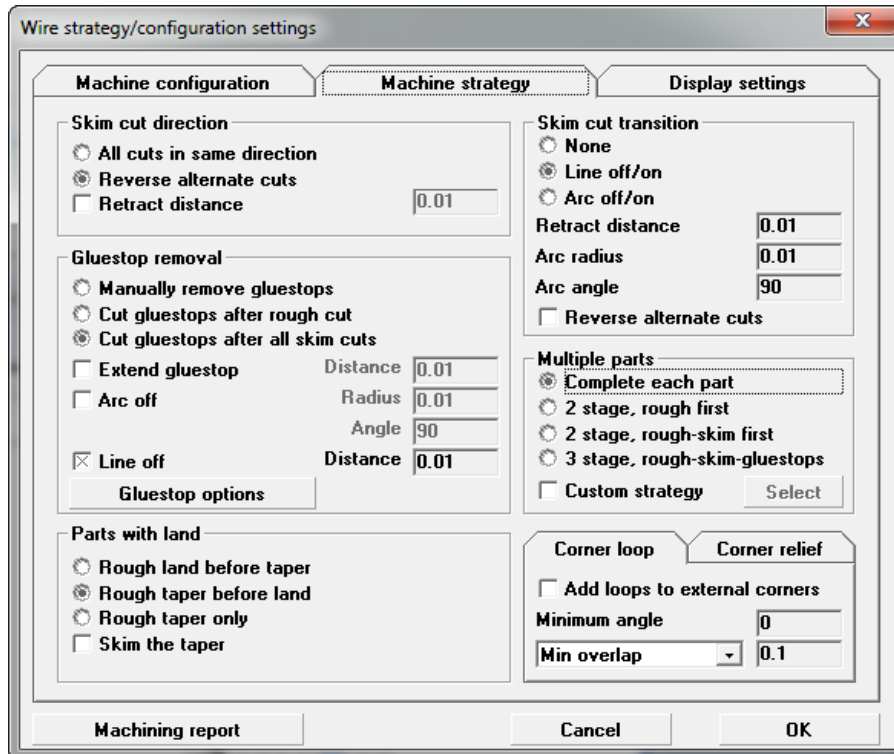
Skim Cut Direction

All cuts in same direction indicates that the roughing cut and all skim cuts will be performed in the same direction around the part. Each cut will start with a wire thread at the start position, lead into the gluestop, and follow around the part until you reach the end of the part. The wire will then be broken so that it can be moved to the start of the next cut. If the part has more than one gluestop, the wire will be broken and re-threaded between each pair of gluestops.

When you select the Reverse alternate cuts option, you have the option of a Retract distance. With this option selected, instead of stopping at the end of a cut and breaking the wire, it reverses back along the profile to perform the next skim cut. The Retract distance option enables the wire to be moved away from the part (normal to the end of the element) so that a new offset can be selected. The wire is then moved back onto the part with the new offset before reversing back around the shape. Thus, for a part with a single gluestop, the wire only needs to be broken once.

For multiple gluestops, the roughing cut and all skim cuts will be performed using alternately reversed cuts between each pair of gluestops. Thus, for a part with three gluestops, the part will be machined as follows:

- All cuts will be performed between the first and second gluestop.

- Wire break, then re-threaded at start position for second gluestop.

- All cuts performed between second and third gluestop.

- Wire break, then re-threaded at start position for third gluestop.

- All cuts performed between third and first gluestop.

- Wire break.

Skim Cut Transition

When all skim cuts are performed in the same direction around the part, the wire offset needs to be changed between each cut. This option provides for the wire to be moved away from the part in order to change the offset before proceeding with the next cut. The move away can be either a straight line or a tangential arc. When you select the Reverse alternate cuts instead of stopping at the end of a cut and breaking the wire, it reverses back along the profile to perform the next skim cut. The Retract distance option enables the wire to be moved away from the part (normal to the end of the element) so that a new offset can be selected.

> (i) Please note that transitions on tapered or 4-axis parts can only be made by a line. Arcs are only available for parallel cuts.

Gluestop Removal

There are three options in this area. The first, Manually remove gluestops, simply means that all of the gluestops will be left in place. This option is useful when parts are held in with small micro-joints, so that the parts do not fall out and the job can be run unmanned.

With the Cut gluestops after rough cut option, the wire will stop at each gluestop (ready for the operator to clamp the part), then continue to remove the gluestop, with the final gluestop on the part

being extended by an optional extend distance. The wire will either be left in position on the part at the end of the gluestop, or an optional arc away from the part can be added.

> (i) If you use this option, since all of the gluestops will be removed before any skim cuts, the reverse skim cuts option will be ignored. There is no need to reverse cuts—all skim cuts will follow the same direction.

With the Cut gluestops after all skim cuts option, each cut will leave the gluestops in place. Only when the final skim cut has been performed will the gluestops be removed. When you select this option with the Complete each part wire strategy, the rough and skim cuts are treated as one set of cuts when pairing the cuts together (the wire is reversed after the rough cut, avoiding the need to break and re-thread). If you check the Skim all gluestops box, then the gluestop will be cut with the same number of skim cuts as the part; otherwise, it will be removed by a single roughing cut.

The Gluestop options button opens a settings dialog, which enables you to specify how the gluestops are removed.
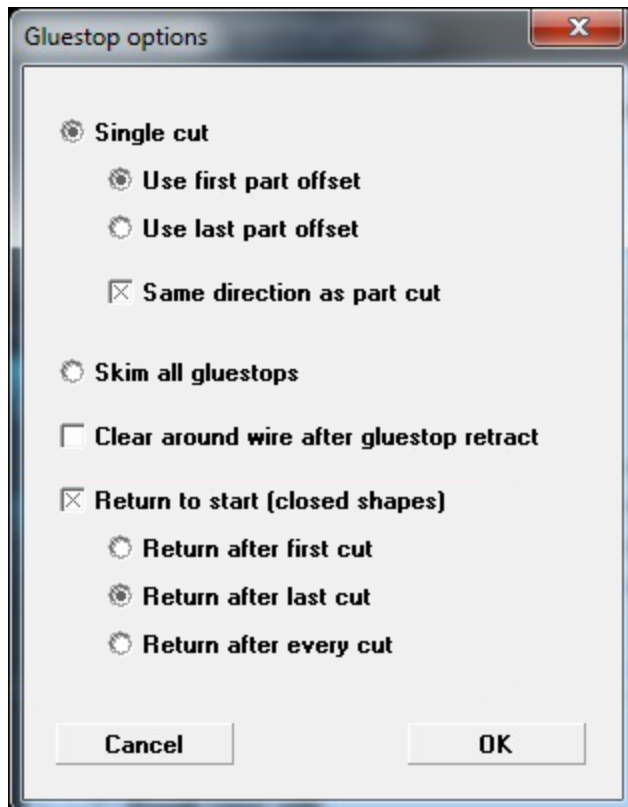
Single cut
A single cut will remove the gluestop, either using the first or the last part offset. If the checkbox Same direction as part cut is enabled, the glue stop will be cut as a continuation of the last pass.

Skim all gluestops
This will make skim cuts along the gluestop using the same number of cuts used on the part.

The checkbox Clear around wire after gluestop retract, when enabled, will make a small square cut around the retract position, to ensure the wire is not snagged in the part.

Return to start checkbox: If the Cut gluestops after all skim cuts option has been chosen (in the main Gluestop removal dialog), and if all skim cuts are in the same direction, the wire can be returned to the start position at the end of the first, last or after every cut. This will be a feed move.

Multiple Parts

The Complete each part option indicates that each part must be roughed and skimmed, and the gluestops removed, before moving onto the next part. 2 Stage, Rough first roughs all parts, then goes back to each part for a second stage of skim cuts and gluestop removal before proceeding to the next part. 2 Stage, Rough-skim first roughs and skims each part, and then goes back to each part for a second stage of cuts consisting of the removal of gluestops. 3 stage, rough-skim-gluestops roughs each part skims each part, and then removes the glues stops from each part.

Selecting the Custom Strategy checkbox will enable the Select button. Click this button to define your individual strategy for machining multiple parts using the dialog shown below.

You will see a list of each operation type (rough, skim, gluestops) which is further divided by part type (parallel, tapered, etc.). Deselect Treat open and closed profiles the same or the Treat punches and dies the same checkbox to get an extended list where parts are further subdivided into open and closed shapes as shown.
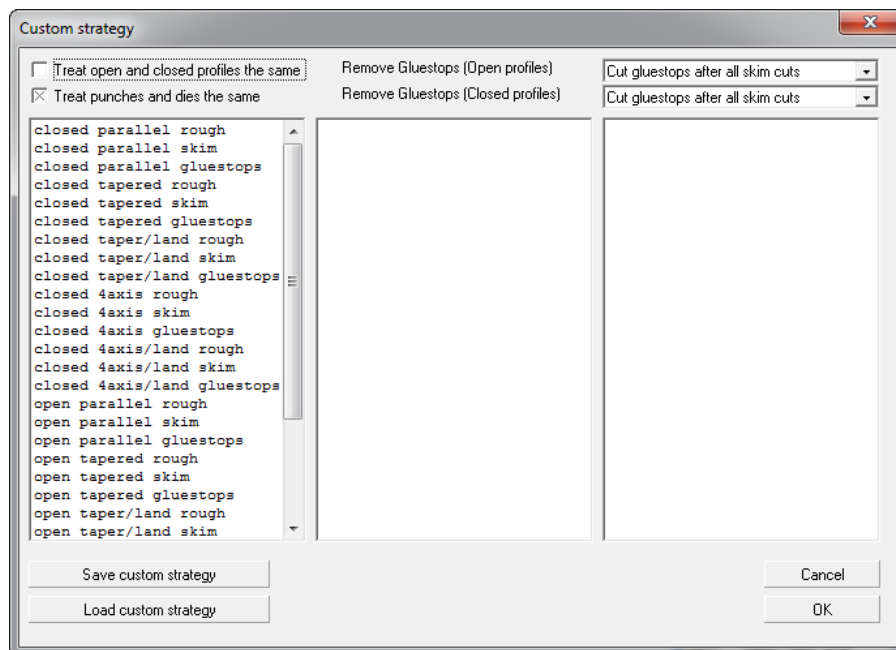
Three options are available for removal of gluestops: Cut gluestops after all skim cuts, Cut gluestops after all rough cuts, and Manually remove gluestops for Open profiles, Dies, or Punches. These gluestop options are fully discussed in Gluestop Removal.

A custom strategy defines how to machine multiple parts using up to eight separate passes across the entire machining process. All of the operations to be undertaken for the first pass will be listed in the first (left-hand) box. To move an operation to the next pass, simply click it to highlight it, then

press the left or right arrow key to move it to the previous or following passes. This example uses four defined passes: The first pass will have four operations; the second pass will have two operations; the third will have only a parallel skim cut; and so forth.
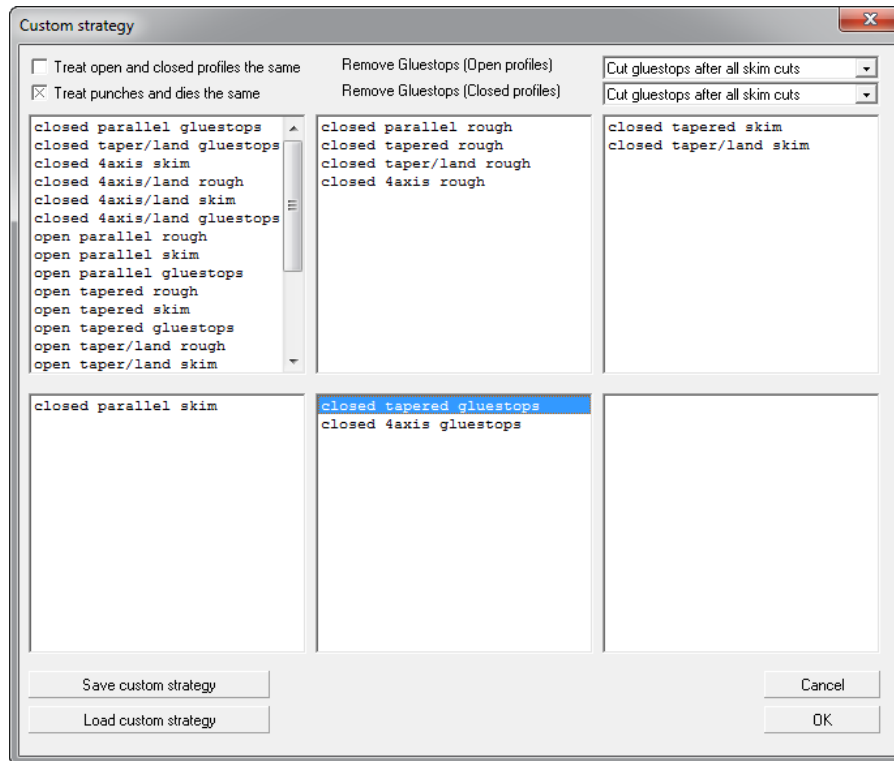
You can save or load your custom strategies for use on other jobs by using the buttons at the bottom of the dialog.

Parts with Land

These option buttons define how land can be handled. The options include: Roughing the land before the taper; roughing the taper first; or roughing the taper only (ignoring the land). Either select Skim the taper to skim both the land and the taper for parts with land, or else deselect this option to skim the land only.

Corner Loop and Corner Relief

These two mini-tabs enable automatic corner looping and corner relief moves. They work in similar fashion, but independently: You can choose to use either, both, or neither of the options. Corner looping and corner relief are available for nocore parts as well.

When Add loops to external corners is enabled, all parallel cuts (including the land on parts with land) will have corner loops automatically added to external corners (including external corners inside dies). A loop will be added by extending the two elements by at least the distance specified (which can be a multiple of the wire diameter or a specific value, selected by using the pull-down menu). Having extended the elements, a tangential arc will be added from the end of the first element to the

start of the second. Note that the radius of the loop will be automatically calculated and will be larger for sharper corners. The Minimum angle setting is used to enter the smallest change in angles between elements before a loop is added.

Three options are offered for corner relief: Do not relieve internal corners, Add loops to internal corners, and Add lines to internal corners. Loops are added by moving back along each element by the specified distance (again, either a multiple of the wire diameter or a specific value) and adding an arc whose center is located at the original corner position. Lines are added at the corner itself, one away from the corner and another back along the same path to return to the corner. These lines will follow a path that bisects the angle of the corner; the length of each move is the distance specified (same as loop radius).

Wire Configuration - Display Settings Tab

The third tab in this dialog provides some control on how the parts are drawn.

Display Settings

Since each skim cut follows the same path as the first roughing cut, if they were all drawn on top of each other, you would have no idea how many cuts had been programmed. Therefore, each skim cut is displayed above the previous cut, with the Z distance between each skim cut being set on this form.

Four-axis parts are drawn with a series of lines between the top and bottom profile to indicate how the two profiles are joined together and synchronized. The distance between these 4-axis lines is also set on this form using Distance between 4 axis display lines.

Show / Hide

You can optionally switch off the display of skim cuts, 4-axis lines and sync lines that have been interactively added to 4-axis parts. By pressing the button to the left of the colored boxes, you are also able to change the colors used to display the lines used for 4-axis display. This is particularly useful for differentiating between system-defined syncs and user-defined syncs.

Wire Guide Display

Choose your preferences for the Wire guide display; you may prefer to display them as Rendered (small), or Rendered (large).

Extend wire through guides is used to provide a more realistic simulation. The wire will be drawn through and past the guides.

Simulation Speed

Adjust the Simulation speed during rendering using the slide control in this section.

Viewport Selection

Under Viewport selection, choose Mouse click to select a particular viewport by moving the mouse over the view and clicking on it. Choose Mouse over to automatically select the view that the mouse is over.

> Mouse over makes it much faster to move between views and to use the mouse zoom and/or pan to get the display you want. However, should you wish to change a particular view (say, plan view to side view), make sure that you move the mouse out of the view and up to the icon to select the new view at the top of the screen without going through another view (since you will simply select a new view as the mouse moves over it). If this is something that you do more often, you may prefer to select Mouse click instead of Mouse over.

Display Datums

Under the Display datums section, you have the option of displaying datum symbols at the XY position defined together with the name of any datum associated with a particular shape.

Agie machines

This version includes Agie Evolution, Excellence and Progress post processors. When one of these posts are selected, a new button, 'AgieCut Settings,' will be available on the first page of Wire Strategy/Configuration Settings.

The Agie post processors require the selection of Material type, Wire type and Machining quality. This selection is displayed by clicking the 'Agie Settings' button which opens the new input form shown below.

Quality

The surface quality of a whole workpiece. The quality of machining is done by these five parameters.

Ra
Roughness.

Te
Corner precision.

Tkm
Tolerance of form.

Speed high
Priority for speed.

Quality high
Priority for high quality.

Output Taper Parts As

There are two different taper modes:

Agie Standard Conic
Taper value is constant on the whole contour (cylindrical).

4 Axis Geometry
The standard 4-axis taper (non-cylindrical).

Material

Material of the pieces that are defined in a technological database. The default material is "Cold die Steel."

Wire

Wires are defined in a technological database insie the AGIEVISION. The default wire is "Cobra Cut A 0.25."

Entry / Exit Start Holes / Minimum Dia (num)

The minimum diameter of all Entry and Exit Start Holes.

Having selected an Agie post, you will be offered a new machining type for closed contours, that of collar machining which is specific to the Agie machine. By checking the new Collar option, you will be offered a new set of parameters that enable you to define a collar part.
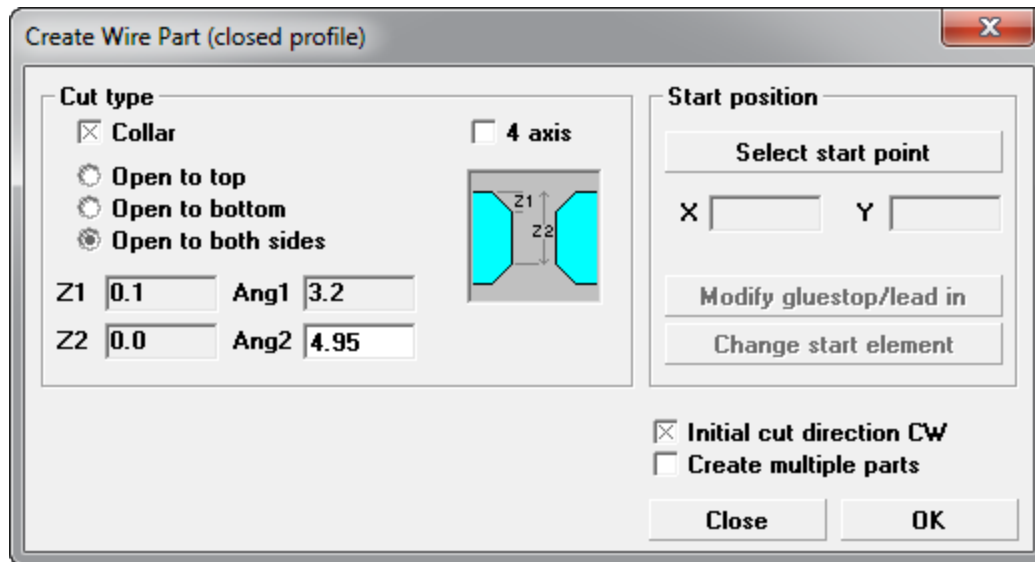
If you select the Open to the top option, you will be asked for the Z1 and Ang1 values, in order to define the chamfer at the top of the part. The Open to the bottom option will also ask for Z1 and Ang1 values, this time used to define the chamfer at the bottom of the collar.

The third option Open to both sides (as shown above will ask for values to define both chamfers. Angles are in degrees and Z1, Z2 are both measured from the top of the part.

Note: In this version, you will see the resulting part displayed as a parallel part (no chamfers shown), but the required collar definition will be output in the NC code.

AGIE Output

When you create code using an Agie post processor, the NC output filename you select is used as the basis for all of the output files created.

For example, if you provide a filename of c: \Evolution\ABC, the following files will be created (all in the c:\ABC folder):

	The job files
`ABC.SBR` and `ABC.SBL`	
	For each part (contour) you will get 2 files

Geometry files

`ABCn.ISO` and `ABCn.ISR`

Where n is the part number (1 for the first part, 2 for second and so on).

For each no-core part you will get 4 files:

`ABCnr.ISO` and `ABCnr.ISR` Geometry files for roughing

Geometry files for skim cuts

`ABCns.ISO` and `ABCns ISR`

For example, for a job with 2 parts and 1 no-core circle, you will get the following files (for the input filename ABC):

`ABC.SBR` and `ABC.SBL` Job files

`ABC1.ISO` and `ABC1.ISR` Part 1 (contour)

`ABC2r.ISO` and
`ABC2r.ISR` Part 2 (no-core roughing)

`ABC2sISO` and `ABC2s ISR` Part 2 (no-core skim)

ⓘ Since the Agie post processor expects filenames to be no more than 8 characters, you must ensure that the filename you specify is no more than 5 characters (files are numbered, so if you have more than 10 parts, you must only use 4 characters.). You must also make certain that you do not add any suffix to a filename.

The Agie post processors require the selection of Material type, Wire type and Machining quality. This selection is displayed by clicking the 'Agie Settings' button which opens the new input form shown on the right.

Quality

The surface quality of a whole workpiece. The quality of machining is done by these five parameters.

Ra
Roughness.

Te
Corner precision.

Tkm
Tolerance of form.

Speed high
Priority for speed.

Quality high
Priority for high quality.

There are two different taper modes

Creating a Wire EDM Part

When you click on this icon and select a profile, you will be presented with the following options
to describe how the part is to be machined.

The first thing you must do is to select a start point. Click on the Select start point button and select a
point that represents where the wire is to be threaded, prior to leading into the start of the part.

If the shape is closed, then the first element to be machined on the part will be the element you selected when you picked the profile. The lead-in will be created normal to this element, starting at the selected point. If the shape is an open profile, you have the option of starting at the start of the first element in the profile, or starting away from the profile (you will need to select a start point).

> (i) If you choose to start on the profile, then you can add entry/exit moves (to apply/cancel compensation) which will take the form of straight line moves either tangential to the profile, or at 90 degrees. You can also opt to extend the start/end of the end elements. If you select a gluestop, this will always be added at the end of the profile (before any extension is added).

Having selected your start point, you must then specify the gluestop width. This is done in the Gluestop lead in/out dialog, shown below. This opens once a point is selected.

You can also enter the diameter of the start hole. This value will be available to the post processor in the event that you would like the approach move split in accordance with this diameter so that you might use different Epak settings for approaching the edge of the hole and approaching the part itself. If you select a circle for the start hole, the circle diameter will automatically be used for the start hole diameter. The start point coordinates are displayed for informational purposes only. These values can only be changed by pressing the Start point button and selecting a new point.

Once you press OK you will be returned to the previous form to specify how the part is to be cut. You will see that the form has changed slightly, so that you can modify this start gluestop.

The Modify gluestop/lead in button is now enabled, and clicking it will bring back the Gluestop lead in/out dialog to enable you to change either the start position, start hole diameter, or the gluestop width.

The Change start element button is also enabled. Clicking here will allow you to select a different element at which to start. This is useful if you picked the wrong element on the profile and would like to start somewhere else on the part.

Having selected your start point, it is a good idea to select the Number of cuts. You have the option of selecting between one and eight. Simply click on the required number at the bottom of the form. Note that this number includes the first roughing cut so, for example, four cuts would equal one roughing and three skim cuts. You may also select which specific wire offsets to use simply by checking the appropriate boxes under Select cuts.

Cut type

The Parallel cut option needs no further data. The wire is simply kept vertical to produce parallel walls. Selecting Full height taper requires a Taper angle (in degrees). The part will always be assumed to be larger at the bottom than at the top, so if this is not the case, you will need to enter a negative angle.

You can change the angle of individual elements of a 2-axis tapered part using the Taper icon. This is the fourth item in the left column of the Top Level palette and is described in Modify Taper and under Modify Taper Angle.

Adding land to a part requires the Land height to be specified. A part with land will be cut once with a roughing offset at the taper angle, then roughed and skimmed with parallel cuts to machine the land.
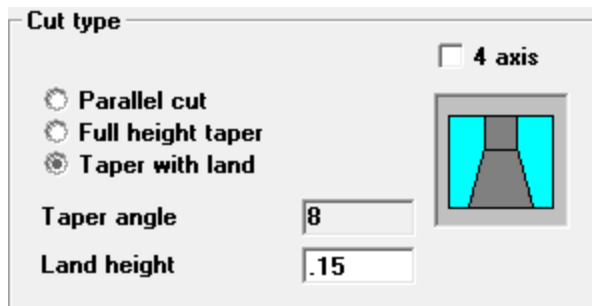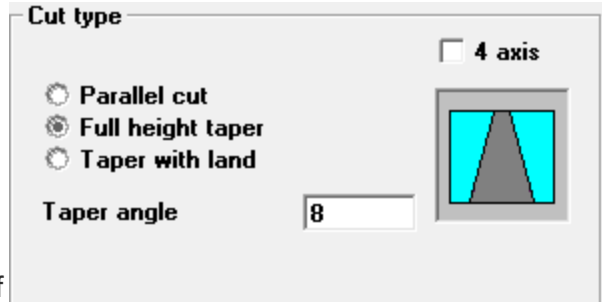
Near the top right-hand side of this section is a checkbox to indicate if this is to be a 4 axis part; checking this will change your options to include Full 4 axis and 4 axis with land.
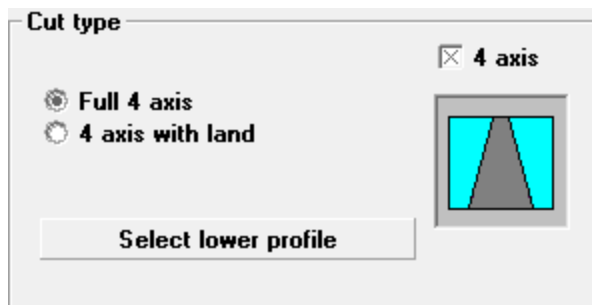
Regardless of whether you choose Full 4 axis or 4 axis with land, you must select a second profile to represent the bottom of the part. To do this, click on the Select lower profile button and then pick the required profile, making sure that the element you select on the profile is the first element to be machined. If you do not select the correct element, the part will appear twisted–this can be corrected later, but it is better to get it correct now.

Having selected the start element on the upper profile, EDM automatically projects an entry line from the start point to the start element and calculates which element on the lower profile this line intersects–this is the default start point on the lower profile. Once you have selected the lower

profile of a 4-axis shape, click on Select start point on lower profile to select a different element at which to start on the lower profile.

> (i) Using the Select start point on lower profile option, EDM will start at the closest end (to the cursor) of this element. This creates an approach move to the 4-axis part that will twist in two separate directions as it moves towards the shape.

The 4 axis with land option requires a Land height and is treated in similar fashion as 2-axis with land. The 4-axis cuts will be performed once, with a roughing offset, followed by one roughing and all skim cuts parallel to the top profile.

Add or Edit a GlueStop

This command enables you to either add another gluestop to a part or to edit an existing one. If you select an element that does not have a gluestop associated with it, you will be able to add a new gluestop.
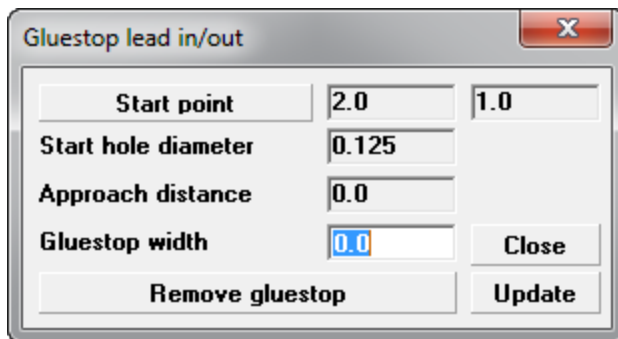
By selecting an element that already has a gluestop associated with it, or by selecting the lead-in to a gluestop, you are able to modify the gluestop definition. You will see when editing a gluestop, you also get the Remove Gluestop button, which allows you to delete a gluestop from a part.

Modify Taper Angle

By selecting this icon, you can then select an element or multiple elements of a 2-axis tapered part for which you can modify the taper angles.
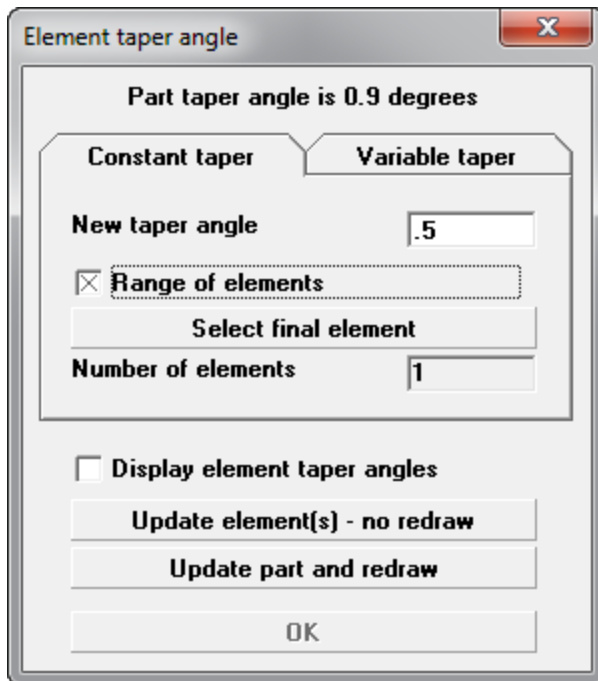
Select the element you wish to modify to get this form.

To modify a single element, enter the value of the New taper angle and click OK. To modify multiple elements, click Select final element and select the element at the other end of the range of elements; the number of selected elements will be indicated in the box underneath this button. Clicking OK will update all the elements in the range with the new taper angle.

Different start and end angles can be specified under the Variable taper tab. Since this can only be applied to one element at a time, the Range of elements option is not provided in this case.
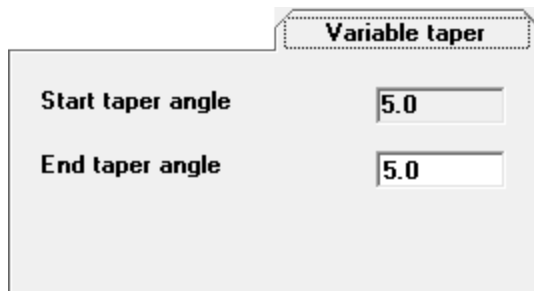
Wire Synchronization

This icon enables you to add a sync line to a 4-axis part. Sync lines are added between the ends of elements on the two profiles that represent the top and bottom of the part. First, select the element on the top profile, picking near to the end where the synchronization is to occur, then click on the corresponding element on the bottom profile.

If you require the wire to remain stopped on one profile, while it continues on the other profile (e.g. stopped on the top and move around a radius on the bottom), then you may add two sync lines to the same element on the profile where the wire is to stop, each of these corresponding to a different element on the other profile. No point may have more than two syncs.
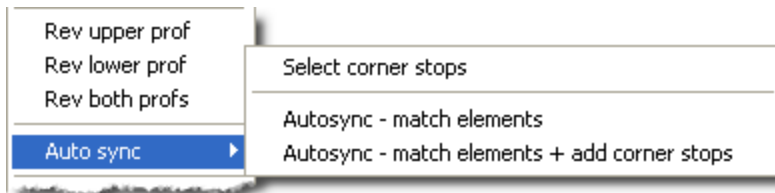

When a 4-axis part has the same number of elements on both the upper and lower profiles, you can right-click on the part and be offered the option to automatically synchronize the cut.

If you have already added user syncs, then this option will further auto-sync between each pair of user syncs.

No-Core Cutting

After selecting this icon, you will be asked to select either a profile or a circle.
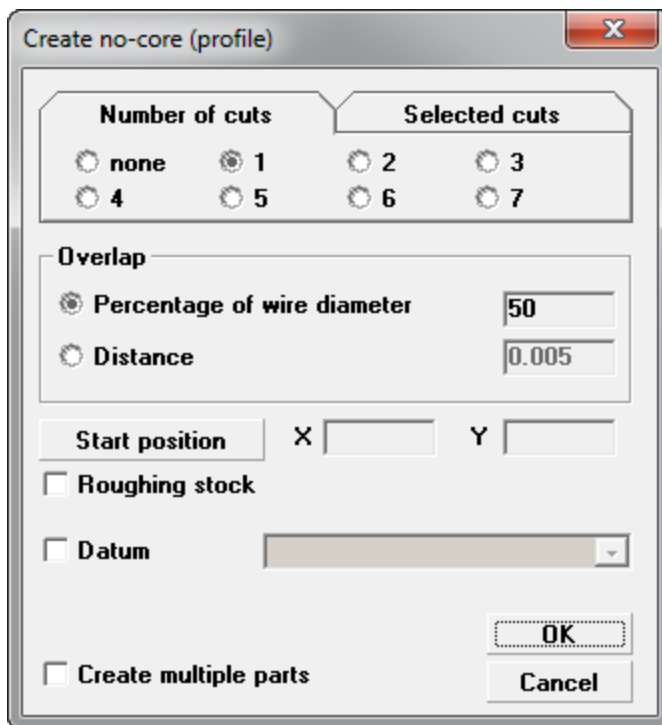
If you select a profile, it must be closed—you will then be able to cut all of the area within the profile by using the dialog shown here. Corner loop and Corner Relief is also available. For more information see Corner Loop and Corner Relief.

Number of Cuts

You can add optional skim cuts around the profile shape by selecting a number from one to eight in the Number of skim cuts tab. You may also select which specific wire offsets to use under the Selected skim cuts tab.

Overlap

The Overlap between cuts can be defined either by a percentage of wire diameter or by an actual distance. If you select Percentage of wire diameter and then change the wire diameter in the wire strategy control, the resulting destruction cuts will automatically be re-calculated based on the new wire size.

Click on the Start position button to select a start point where the wire will be threaded.

If you select a circle for No-Core cutting, then you will be presented with the dialog shown. As for the cutting profile, you can specify the Overlap as either a distance or as a percentage of the wire diameter (automatic re-calculation also applies, should you change the wire diameter).

The start position will be displayed as the center of the selected circle and the Final diameter is also taken from the selected circle. You can change the Initial diameter to be a value other than zero, in which case the No-Core cutting will start at that specified diameter, rather than at the center of the circle.

Checking the CW box will create clockwise arcs. Leaving it unchecked will create counterclockwise cuts.

Dimensioning the Part

This icon offers a palette of icons for dimensioning your part.

The top row of icons enables you to dimension a distance. To use any of the first three, you will be asked to select a line; you will then be shown the dimension of that line in either the horizontal (X) axis, the vertical (Y) axis, or the actual dimension parallel to the line. Move the mouse to position the dimension and then left-click to position it. Right-clicking will abandon the dimension.

The next three buttons are similar, except you must pick two points and the dimension will be the distance between the points (again, either horizontal, vertical, or parallel).

On the bottom row, from left to right, you can dimension a point (the XY position), the angle of one line, the angle between two lines, the radius of an arc and the included angle of an arc. The last icon on the bottom row is used to configure the dimensions.

Using this dialog, you will be able to specify the number of decimal places to be used for both linear and angular dimensions and also the height (in inches) of the dimension text.

> (i) The dimension text height is used for all subsequently defined dimensions; existing dimensions will not be changed to reflect this new height.
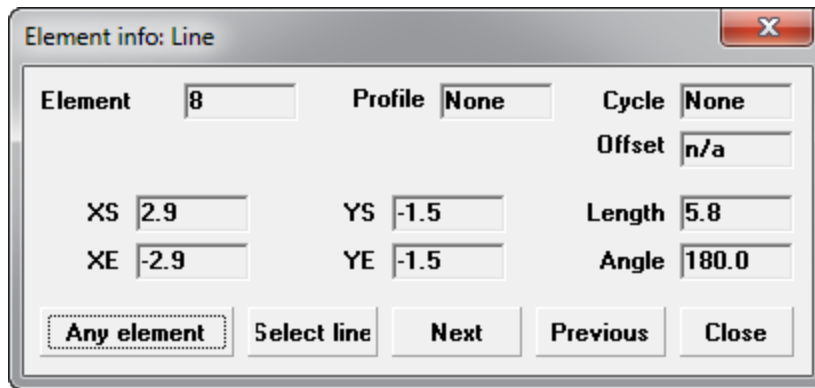
Information

This icon offers several methods of obtaining information relating to the part via the following palette of icons.


The first two icons provide information on a selected element. Use the first icon to select a construction element (point or circle).
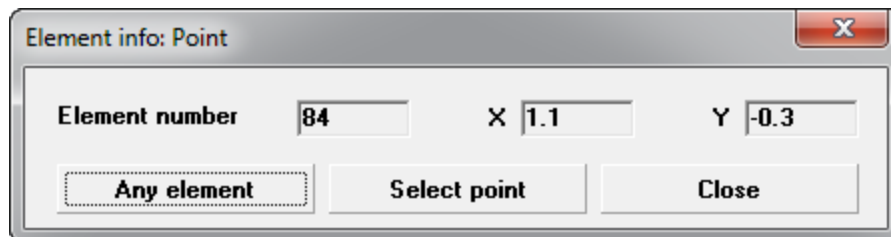

The second icon is for a profile line or arc. Click on the icon and then select the required element and you will be shown information specific to the type of element selected. This may include start/end point, length, and angle.

Element info: Line

Element	8	Profile	None
			Cycle None
			Offset n/a
XS 2.9		YS -1.5	Length 5.8
XE -2.9		YE -1.5	Angle 180.0

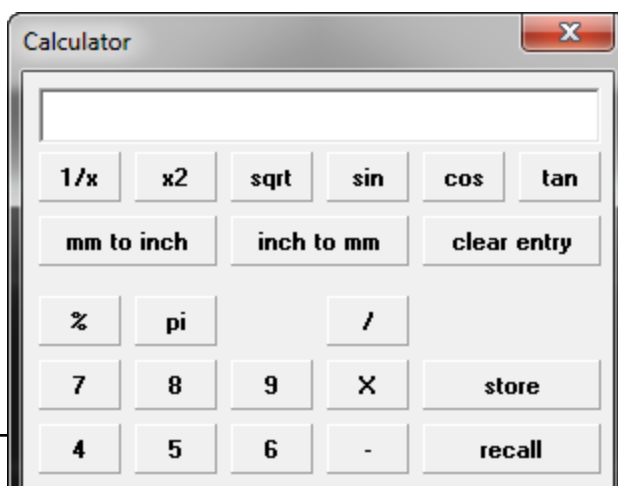Any element   Select line   Next   Previous   Close

The third icon is used to measure the distance between two points. After selecting the two points, you will be shown the distance between them relative to the X axis and the Y axis as well as the actual distance between them.

GibbsCAM Wire EDM, v10

Distance in X =0.0
Distance in Y =0.0
Distance =0.0

OK
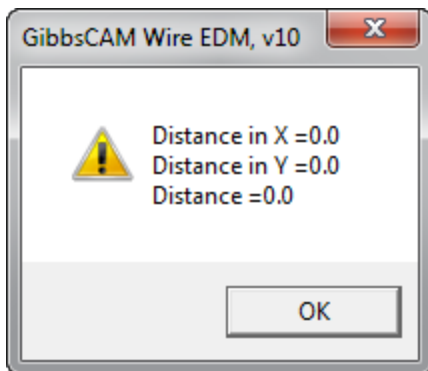
The fourth icon will ask you to select a profile; this will calculate the area and peripheral length around the profile.

GibbsCAM Wire EDM, v10

Area =14.7854
Perimiter =15.1416

OK
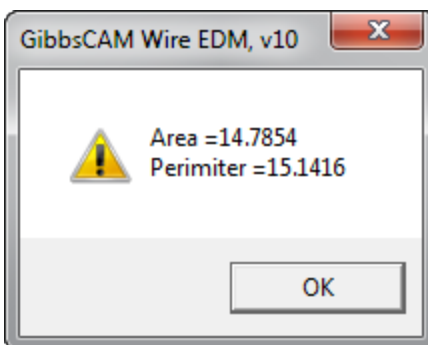
The last icon provides a desktop calculator that includes memory/recall and trig functions.

Calculator

1/x	x2	sqrt	sin	cos	tan
mm to inch		inch to mm		clear entry	
%	pi		/		
7	8	9	X	store	
4	5	6	-	recall	

Resequence the Operations

The order in which the machining occurs is determined by the order in which you create wire parts. If you have more than one part, then the first one you create will be machined first. Using this icon enables you to change the machining sequence.
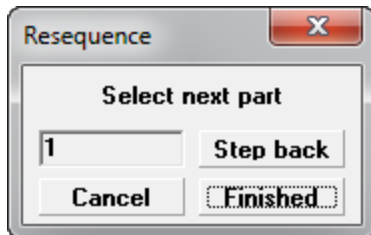
You will see a small form that keeps track of the number of parts you pick. Simply click on the part that you want to cut first, followed by the next one and so on until you have selected all the parts. Then click the Finished button and the machining will be resequenced.

> ⓘ This option only enables parts to be resequenced; it will not affect the order of machining No-Core cuts, which will always be cut in the order in which you define them. Destruction cuts will also occur before cutting any wire parts.

Machining Simulation

You can see a simulation of the toolpath by clicking on this icon.
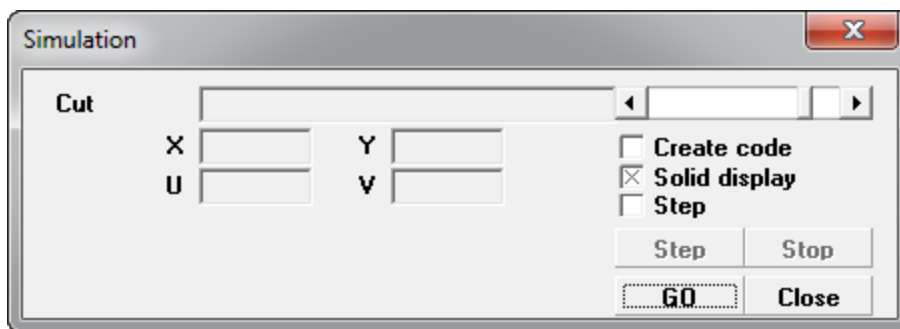
The form displayed offers a horizontal scroll bar to select the speed of simulation and a Step option to stop the wire at the end of each line or arc. If you select the Step checkbox, then the wire will stop at the end of each element. You will need to click the Step button repeatedly to proceed through the elements.

> ⓘ Moving the scroll bar to its maximum value will cause the wire to be displayed at the end of each element only. You will not see the wire move along any of the elements when this speed is selected.

Creating CNC Code

This icon provides the same toolpath simulation as described above, but CNC code is created as the machining is being simulated. You will see a list of post processors that you have available and also a box containing the file name of the CNC code file. To select a new code filename, simply double-click on the box containing the filename. As the simulation is displayed, you will also see the code being created.
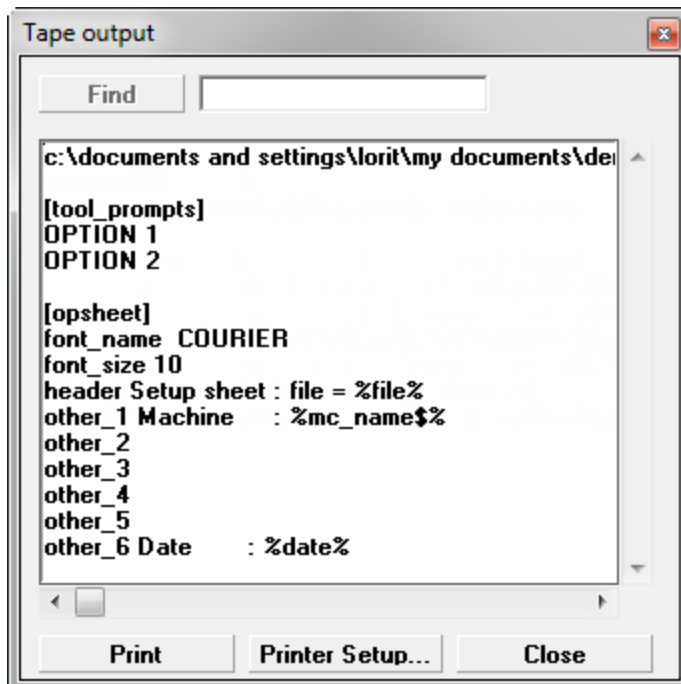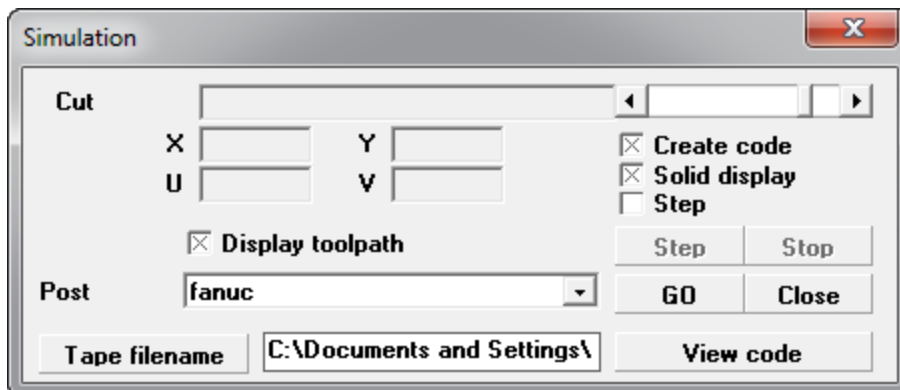
Right-Click Editing

As an alternative to using the icons to modify a part, you can simply point to a line or arc on the screen and perform a right-click. Depending on what type of element you click on, you will be offered a small pull-down menu that provides various options similar to those shown. The menu on the right is shown when you pick a wire part (not a No-Core cycle).

The first line is an indication of what type of part has been selected. This will be set to one of three options, Punch, Die or Open Profile.

The next option Line: info indicates that the element selected is a line. Selecting this option will show you information about the selected line, including line length, angle, start and end position.

The number of cuts and cut length are for informational purposes only. The cut length is the length of a cut around the profile.

The Rev prof dir option provides a means of reversing the profile direction; clicking this will reverse the profile and will cause all of the machining to be automatically re-calculated. The context menu for a 4-axis profile, on the other hand, will contain Rev upper prof and Rev lower prof as well as Rev both profs.

Clicking on the Delete machining option will delete all destruction cuts.

Pattern repeat offers the ability to repeat the selected machining along a path. To apply a pattern, check the box to use pattern and press the select path button, then pick a contour. You will be shown how many repeats will be created as a number on the form; press OK to apply the pattern. To remove the pattern repeat, right-click again and un-check use pattern box and press OK. The path must be a contour, where the machining will be repeated at each point along the contour (each connector / terminator). The toolpath will be translated so that first wire thread position in the machining will be located at each point on the path.

The Properties option at the bottom of the menu provides a method for updating the selected part. If you picked a line that represents a lead-in to a gluestop, then the clicking on Properties will show the form containing the gluestop data. Any other element of the part will show the form containing the part definition data. In either case, you will be able to change the data and update the part.
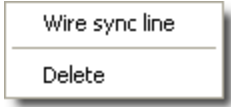
If you right-click on part of a wire No-Core cycle, you will get a slightly different menu.

The first line indicates whether you selected a circle or a profile. The third line provides access to element information. The cut length is the total peripheral length of all the cuts used to destroy the circle /shape. Clicking on the Delete machining option will delete all destruction cuts.

Clicking on the Properties option will show the form that was used to create the cycle so that you can change the machining parameters.
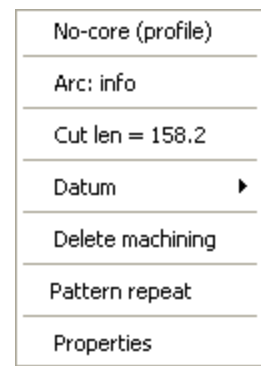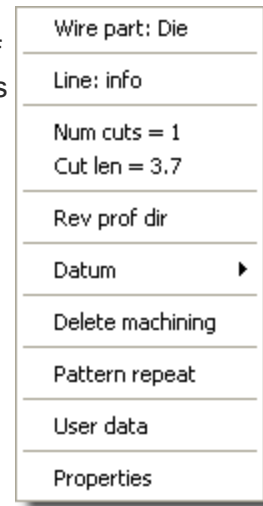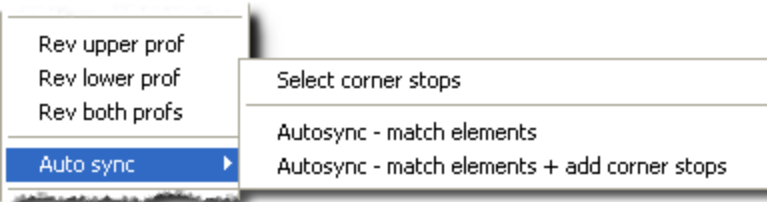
Right-clicking on a 4-axis sync line provides a menu option to delete the sync line.

If you Right Click on a 4-axis line or arc or on a No-Core cycle, you will get the menu shown below.

If you have already added user syncs, then this option will further auto-sync between each pair of user syncs.

If you right click on any other type of element (a point, a dimension, or a line/arc that has not been used to create a wire part), you will be presented with a menu that simply offers element information (in the case of a dimension, you can also delete the selected dimension).

Select the (Element): info option to bring up the Element info dialog as shown.

Post Processing Enhancements

A new post processor ('drill.pp') has been added to the library of posts. This will create a file containing only XY positions of the wire thread positions to be used to drill the material prior to use on the EDM machine.

A new document has been provided to help those wishing to modify or write their own post processors. This will include details on new post processing functionality added to this release:

- Simpler commands to format the output, thereby enabling easy switching between inch/metric and absolute/incremental output.

- Current single-line IF command enhanced to include IF/THEN blocks and IF/ELSE.

- The ability to output part profile data (rather than toolpath data) as required by machines such as the Agie Evolution.

- Added flag in posts to indicate if part is upside-down.

- Added new post variables for Part name, Part Filename, Tape Name, and variables for the new datum functionality.

Creating your own posts
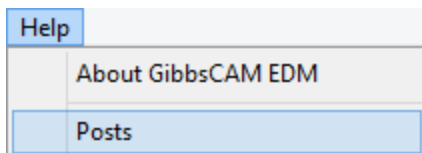
In the Help menu there is an entry called Posts. This item contains information that will help those wishing to modify or write their own post processors.
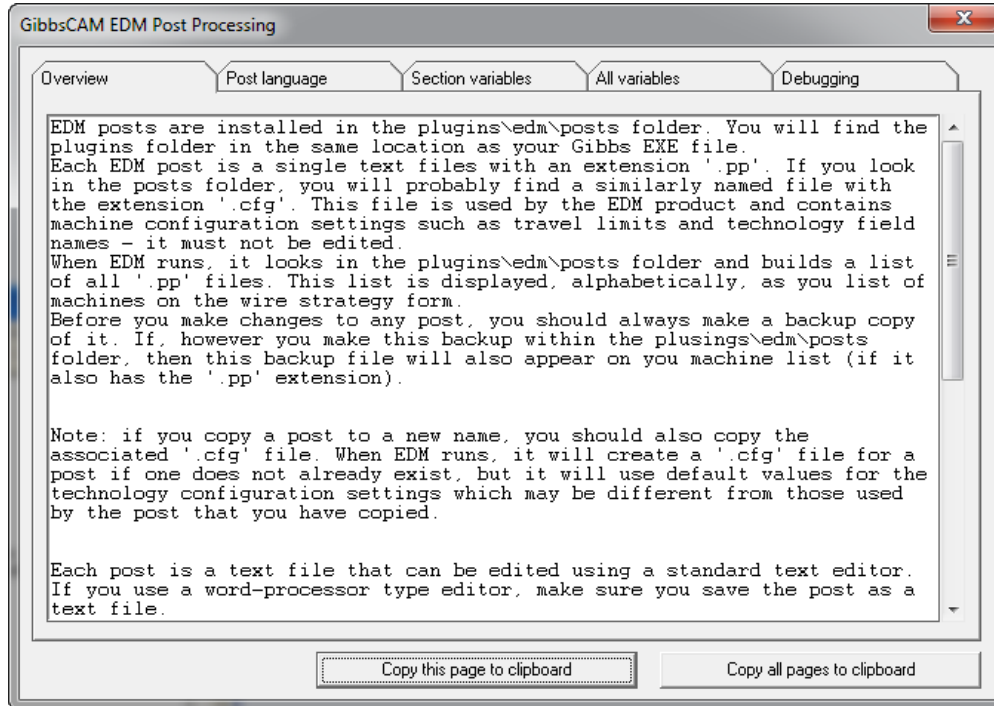
This includes details on new Post Processing functionality in the Wire-EDM module including:

- Simpler commands to format the output, thereby enabling easy switching between inch/metric and absolute/incremental output.

- Current single-line IF command enhanced to include IF/THEN blocks and IF/ELSE.

- The ability to output part profile data (rather than toolpath data) as required by machines such as the Agie Evolution.

- A flag in posts to indicate if part is upside-down.

- New post variables for Part name, Part Filename, Tape name and variables for the new datum functionality.

About creating your own posts

The default EDM posts are installed in the user data folder, default
`C:\ProgramData\CAMBRIO\GibbsCAM\<version>\EDM\Posts\`. You will find the plug-ins folder in the same location as your Gibbs EXE file. Each EDM post is a single text file with the extension ".pp". Also in the `Posts` folder there should also be a similarly named file with the extension ".cfg". this file is used by the EDM product and contains machine configuration settings such as travel limits and technology field names. *This file must not be edited*.

When EDM runs, it looks in the `EDM\Posts` folder and builds a list of all ".pp" files. This list is displayed, alphabetically, as the list of machines on the wire strategy form.

Before making changes to any post, be sure to always make a backup copy of the post. If this backup is made within the `EDM\Posts` folder and has the ".`pp`" extension it will also appear on the machine list.

> ⓘ   If you copy a post to a new name, you should also copy the associated ".`cfg`" file. When EDM runs, it will create a ".`cfg`" file for post if one does not already exist, but it will use default values for the technology configuration settings which may be different from those used by the post that you have copied.

Post Files

Each post is a text file that can be edited using a standard text editor. If a word-processor type editor is used, make sure to save the post as a text file. The post file is not case-dependent, although lowercase is generally used.

Comments and Variables

Comments may be included by preceding the comment with an exclamation mark (!). Comments can either be added to the end of a line or a complete line may be a comment. Where a numeric value is to be used, an expression may be used. Your own variables may be also be created to help with calculations. Variable names may be up to 32 characters. Variable names must start with a character (A-Z), can contain the underscore character ("_") and numbers (0-9).

To avoid conflicts, use names that are not likely to be used in future versions of the software when defining variable names. This can be done by adding a user-specific prefix. For example, instead of "offset1", use "post1_offset1".

Valid Variable Definitions

- `x1=10`

- `x2+20 * Cos(45)`

- `x3=((x1+2/2)+(x1*x2/2.9)**2)*sin(45)`

All variables are treated as floating point numbers. There are several text variables that may be used to create output. You cannot create your own text variables and there are currently no text manipulation functions.

Drilling

The post processor "`drill.pp`" may be found in the library of posts. This will create a file containing only XY positions of the wire thread positions to be used to drill the material prior to use on the EDM machine.

Conventions

GibbsCAM documentation uses two special fonts to represent screen text and `keystrokes or mouse actions`. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

Text

Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A "torn" edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:
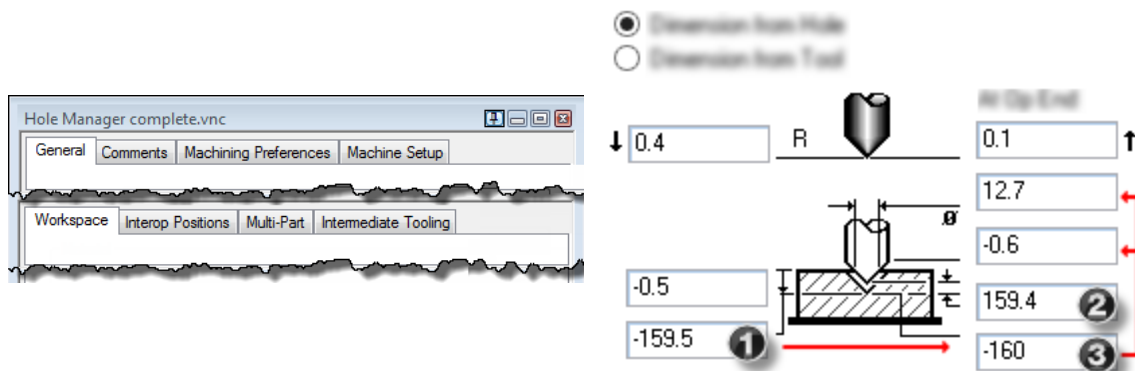
Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

Faint green borders that outline areas within a graphic usually signify an image map. In online help or a PDF viewer, you can click a green-bordered area to follow the link.

Links to Online Resources

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens a restricted website containing materials available for download. Requires a GibbsCAM Online Services account; to set up an account, contact GibbsCAM Support.
Go	https://store.GibbsCAM.com	Opens the website for the GibbsCAM Student Store.
Go	https://macros.gibbscam.com	Opens a wiki containing documentation and examples of GibbsCAM macros. Requires a GibbsCAM account.
Go	http://kb01.GibbsCAM.com	Opens a Knowledge Base article, **Contour Operations Using Thread Mill Tools**, that explains in detail the correct way to program Contour processes using Thread Mill tools.
Go	mailto:Support@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Technical Support department for GibbsCAM.
Go	mailto:Registration@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Registration department for GibbsCAM.
Go	mailto:Sales@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Sales department for GibbsCAM.
Go	http://www.autodesk.com/inventor	Opens an external website that provides more information on Autodesk Inventor products.
Go	http://www.celeritive.com	Opens an external website that provides more information on VoluMill Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.
Go	http://www.predator-software.com	Opens an external website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

Index