CutViewer Mill V3.2

Introduction

What is CutViewer Mill?

CutViewer Mill V3.2 is an easy to use program that graphically displays the material removal process for milling/drilling operations in 2, 2.5, or 3 axes. Based on the stock statements and tool definitions, CutViewer will show you exactly what material will be removed from a raw stock.

What Hardware does CutViewer Require?

CutViewer is a 32 bit application. It runs in Windows 95/98/ME/NT/2000/XP. The hardware minimums include a Pentium processor, 64 MB of RAM, 100 MB of hard disk space, and a video graphics card with at least 2 MB of video RAM.

Display Settings

Before running CutViewer, set the Windows Color Palette to 16-bit mode (65535 colors) or higher.

Evaluation Version

The CutViewer download is the full version with a 30 day time limit. After that time a license serial number is required to continue using the software. To obtain a license go to http://www.cutviewer.com

CutViewer Features

CutViewer can open and display Standard ISO G-code files. We can also provide upon request, a special .dll file that enables it to read any control. CutViewer provides easy real-time manual editing and verification of the program file, viewing the process from different viewpoints of the part, measuring the finished part, collision detection, and many other features.
The post reader can be user customized for your specific CNC hardware.

The raw stock can be defined as either a squared block or a cylinder. Stock and tool data can be included in your CNC program, or defined separately prior to running the program.

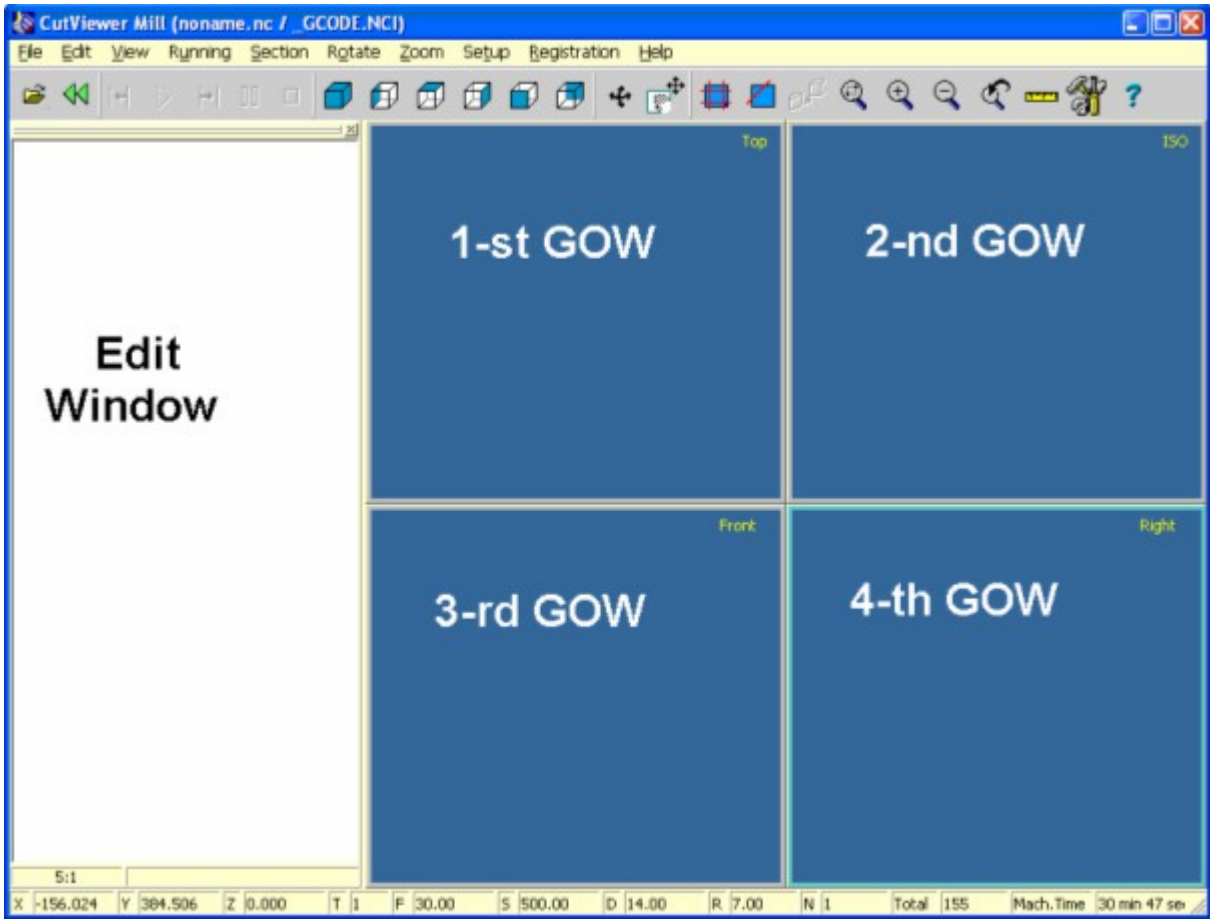
This version supports the following types of tools:

- Ball End Mill
- Bull End Mill
- Flat End Mill
- Corner Rounding End Mill
- Tapered Mills
- Chamfer Tools
- Drills
- Center Drills

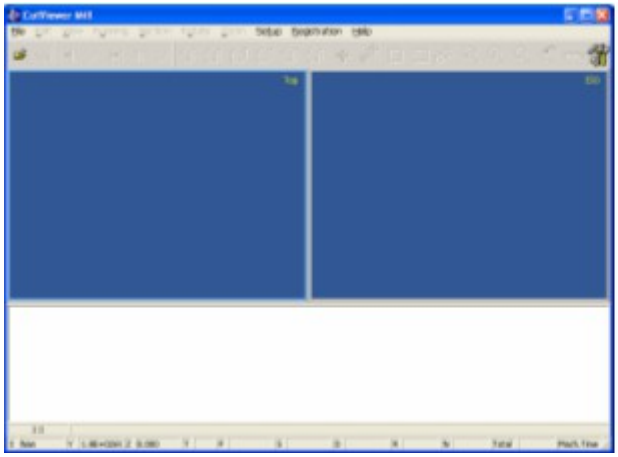
The ball mill tool can be used with control points on either the ball surface or on the ball center.

Running CutViewer

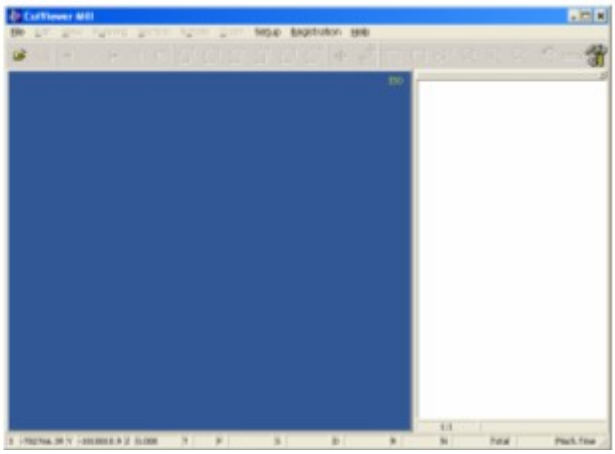
When you first open CutViewer it appears as a reduced size window. You can maximize the main window but take into account that window size affects the run speed.
Screen layout contains the Edit Window and 1, 2 or 4 Graphics Output Windows (GOW). The GOW's are the graphical output areas for simulation, i.e. the views. Up to four different views can be represented at the same time. The graphic window with the focus is identified by blue borders.



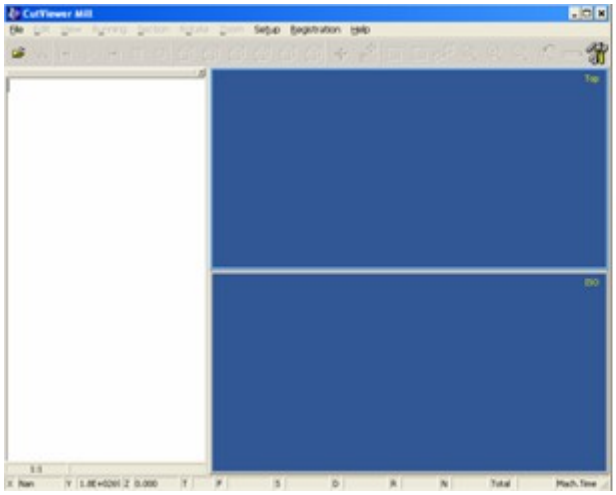
You can change the number of GOW's (1, 2, or 4), the orientation of GOW's (Vertical or Horizontal) and Edit window position (Top, Bottom, Left, Right) via Setup mode. The pictures below show some possible screen layouts.



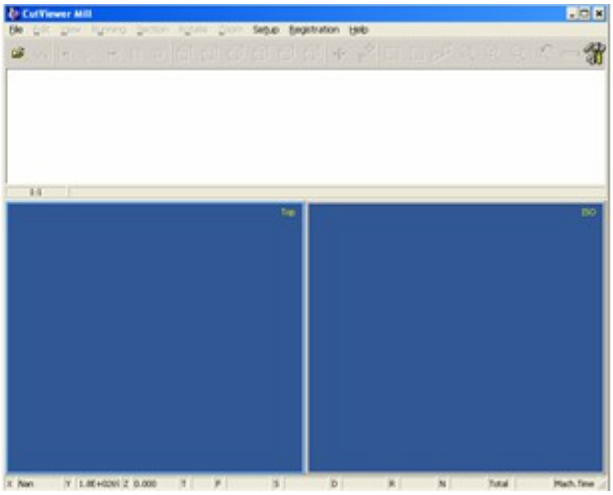
Edit Window Position=Bottom
Number of views=2
GOW Orientation=Vertical



Edit Window Position=Right
Number of views=1

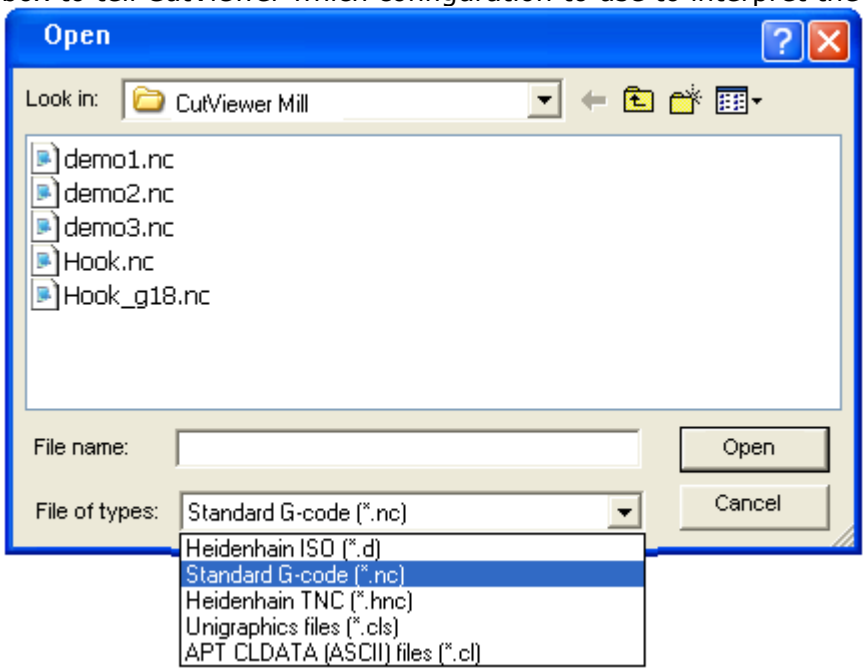


Edit Window Position=Left
Number of views=2
GOW Orientation=Horizontal



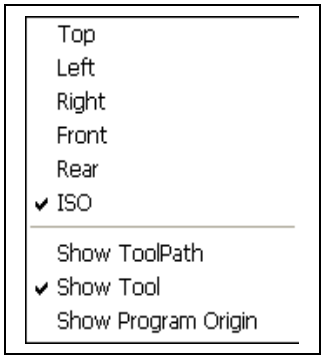
Edit Window Position=Top
Number of views=2
GOW Orientation=Vertical

After starting CutViewer only the File-New, File-Open, and Setup items are available. All windows are empty when they appear. To load a file, select the Open File button and navigate to the desired directory. Select the appropriate filter in the «File of types» combo-box to tell CutViewer which configuration to use to interpret the g-code.



*Important: selecting wrong configuration causes incorrect simulation (e.g. if you open Fanuc NC file selecting Heidenhain configuration).

If stock and tool information has not been added to the NC post file before hand, a series of dialogs will prompt you to define them when the NC file is loaded. CutViewer maintains tool database files in the CutViewer directory for each type of tool you choose to save there. These tool database files have a *.db3 extension. You can store and retrieve these tools for use in all of your simulations.
When finished, the Editor window will display the NC file and a shaded Stock will be shown in the Graphic Windows.
Each graphic window has its own settings. You can change these settings through the window popup menu (in order to open this menu the user should click with the mouse right button somewhere on the window area).



The popup menu for graphic windows provides the possibility of making window-related settings.
One of the following views can be selected for each graphic window:
Top
Left
Right
Front
Rear
ISO
The «Show Tool path» menu item activates and deactivates the output of the line graphics for the toolpath. The «Show Toolpath» command will not take effect until the simulation has been started or one of the view items has been selected.
The «Show Tool» menu item activates and deactivates the output of the tool graphics.
The «Show Program Origin» menu item activates and deactivates the output of the Program Origin graphics.
The current state of the window settings is indicated by the checkmark at the appropriate menu item.

Press the «Run» button to start the simulation. To make the on screen cutter speed slower you can define «Dwell» value (milliseconds) in «Setup» mode and the process will be suspended for the specified dwell time at every screen point of the tool motion.
To make the cutter move faster during processing simply press '+' key
and press the '-' key to slow it down. Press «PgUp» key for the fastest cutter speed.

There are two modes of simulation accessible through Setup:
1. Solid: Graphically simulates the material removal process by updating the stock solid shape as the tool moves (by default).
2. Turbo: Graphically updates only the toolpath with the tool image (if «Show Toolpath» and «Show Tool» were activated). The machined part appears only after reaching the end of NC file or when the user presses «Step Forward», «Pause» or «Stop» button.

If you want to create new NC program directly in the CutViewer select the «File-New» menu item, then select the desired CNC type and an empty Noname.nc file will be loaded to the Edit window. Now you can type the program. To save new program to the disk select the «File-Save as» menu item. To simulate new program select the «Run-Rebuild» menu item or simply press Ctrl-F9. If Tool or Stock info are ommited then CutViewer will initiate the appropriate dialog prompting for missing info.

The following buttons are available at the Main Toolbar:

Open File

This is for opening NC file from a folder or across network. But make sure you select the appropriate filter in the «Files of type» combo-box to tell CutViewer which configuration to use to interpret the g-code.

Reset

Resets the simulation showing the initial Stock at the start position.

Run

Press this button to start the simulation.

Step forward/backward

Switches to step-by-step mode, as this button is pressed repeatedly, the tool moves incrementally forward or backward. When in Step mode, the line being executed will be highlighted in the Edit window and a separate dialog will display the current coordinates and cutting parameters. This helps in identifying the lines to edit. To cancel the incremental trace mode press the «Run» button.

Note: you can simply press the «Space» key to switch to step-by-step mode during execution. Also you can use the «arrow» keys instead of the «Step» buttons when step-by-step mode is active. The tool will move incrementally forward if you press «Down» or «Right» key and the tool will move incrementally backward if you press «Up» or «Left» key. Press «Return» key to cancel the incremental trace mode and continue execution or Press «Esc» key to stop execution.

Pause

Suspends execution. While processing is paused, you may apply zooming, measure the part as well change the Setup parameters and EDIT the code. The «Run» button will restart simulation from the paused position with provision for new parameters. You can also edit the file, but you have to select the «Run-Rebuild» menu item or press Ctrl-F9 in order to implement the changes.

Stop

Stops execution of the simulation. Unlike «Pause», you cannot restart from the current position after pressing «Stop». Pressing «Run» following «Stop» will start the simulation from the beginning of the file. Also press «Run» following «Stop» if you want to restart process after finish of the simulation. The simulation will be started keeping the latest scale for each graphic window. If you want to start with the initial scale press the Reset button.

View buttons

Press the appropriate button to set the desired view for the graphic window that has a focus (this window is identified by blue border).

'Parallel to Axis' sectioning

Generates a sectioned part slicing it with a plane parallel to the Axis.

'Through 2 points' sectioning

Allows you to cut away a portion of the part through any vertical plane defined by two points.

Unsectioning

Press this button to restore the sectioned part.

Zoom Window

Allows for the zooming by a window or region selection method. To use this function, click the button and while holding down the left mouse button select the desired window and then release the mouse button.

Zoom In

Press this button and then press the left mouse button at the desired point to Zoom in on a part. Press the right mouse button to leave Zoom In mode.

Zoom Out

Press this button and then press the left mouse button to Zoom out on a part. Press the right mouse button to leave Zoom Out mode.

Zoom Restore

Reverts to the initial zoom view.

Pan

Moves the part within the window. Press the button, and then hold the left mouse button down while moving the mouse. When the view is in the desired position release the mouse button. Press the right mouse button to leave Pan mode.

Rotate

Rotates the view. This mode is available for ISO view only. Press the button, then hold the left mouse button down while moving the mouse. When the view is in the desired position release the mouse button. Press the right mouse button to leave Rotate mode.

Measurements

Displays a tool for making measurements of the part.

Setup

Launches a dialog where you can configure how CutViewer works. If you press the save button to exit rather than the OK button your settings will be the default the next time you launch CutViewer.

Tool and stock definition

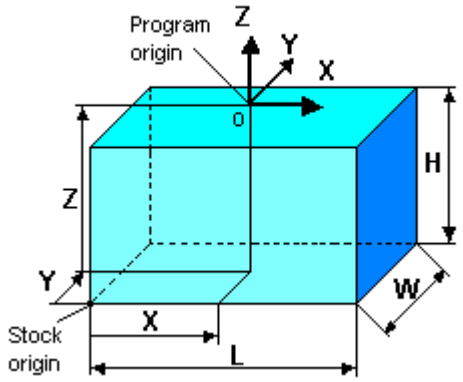
As CutViewer loads an NC file, it looks for information about the stock size, tool geometry and program origin placement relative to the stock origin. This requires the addition of several lines to the NC file. If these lines are omitted from the NC file, CutViewer will initiate a sequence of dialogs prompting for the missing information. If you prefer to insert the lines manually follow the directions below:

Each NC file must contain following additional lines: STOCK, FROM, and TOOL. Tool, Stock and From definition lines should be prefixed with a semicolon, left parenthesis or the character your CNC controller uses for a comment line.

The stock shape may be a Block or Cylinder with a bore.

The line format for a Block is described below:

a) **;STOCK/BLOCK, Length, Width, Height, Origin X, Origin Y, Origin Z**



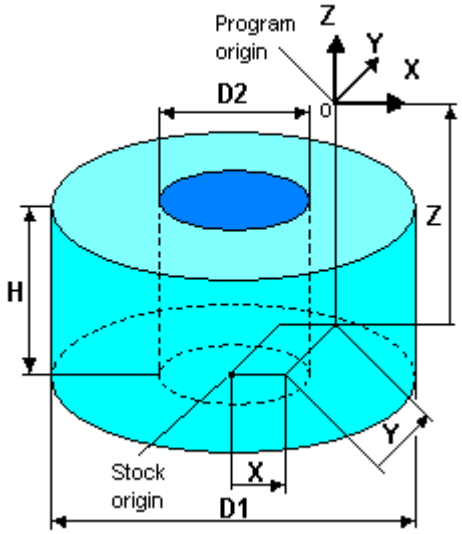
The first three variables are the stock size, given in positive numbers representing the actual measurements of the material. The stock reference point (origin) is the lower left corner in the isometric view (the point with the minimal X,Y,Z coordinates) . The Stock origin also can be identified by the intersection of the yellow, blue and red axes when the rotate (view) button is pushed. The last three numbers indicate the x,y,z position of the program origin relative to the stock origin. This definition provides more control than assigning z=0 at the top surface of the block, it also requires more attention.

For example, if the stock has a length of 100 mm, a width of 70 mm, a height of 30 mm and the program origin is at center of the top face:
;STOCK/BLOCK,100,70,30,50,35,30

Similarly, a 6»x4»x1» stock with a program origin at the center of the top surface is written as:
;STOCK/BLOCK,6,4,1,3,2,1

If the stock shape is a Cylinder with a bore:
;STOCK/CYLINDER, D1, D2, Height, Origin X, Origin Y, Origin Z

D1 is Outer Diameter
D2 is Inner Diameter



The inner diamter D2 may be 0. Diameters and Height represent the actual measurements of the cylinder. The stock reference point (origin) is the center of the cylinder bottom.

For example, if the cylinder has a diameter of 100 mm, a height of 30mm and the program origin is at center of the bottom face:
;STOCK/CYLINDER,100,0,30,0,0,0

If the program origin is at center of the top face:
;STOCK/CYLINDER,100,0,30,0,0,30

if the cylinder has a bore diameter 40 mm:

;STOCK/CYLINDER,100,40,30,0,0,30

If you start the program with the wrong Stock parameters you can re-deine the stock selecting the «Redefine a Stock» menu item from the Edit window popup menu or simply pressing Alt+S and the Stock definition dialog will appear.

b) **;FROM/X start, Y start, Z start**

The FROM line denotes the starting or home position of the tool relative to the program origin. In this example using the stock above and the initial position of the tool 10 mm above the top face:
;FROM/0,0,10

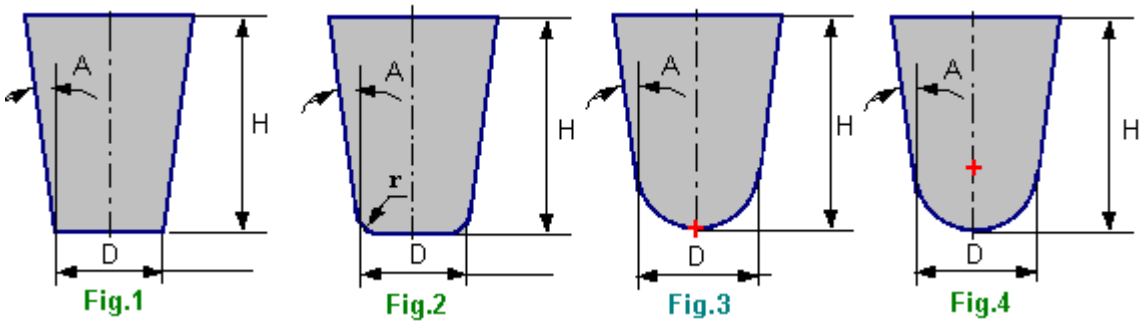
The inch example, if the first tool position is .1» above the origin:
;FROM/0,0,.1

c) The TOOL line denotes the tool geometry.

The line format for Mills is described below:

;TOOL/MILL, Diameter, Corner radius, Height, Taper Angle

*Note: for Tool/Mill: the Corner Radius will indicate the type of mill used where 'r' is the corner radius specified and 'D' is the tool Diameter:



r=0 for a flat end mill (Fig.1)

r<D/2 for bull end mill (fig.2)

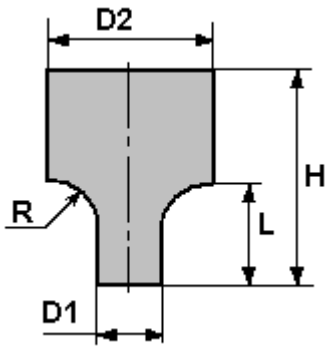
r=D/2 for ball end mill (if the control point is the tool tip) (Fig.3)

r=-D/2 for ball end mill (if control point is the center of the ball) (Fig.4)

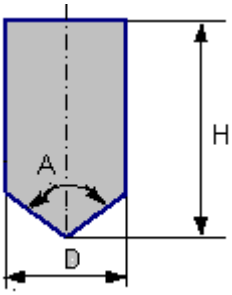
The following are Examples of various tool settings:
;TOOL/MILL,20,0,80,0 ;TOOL/MILL,1,0,1.5 - flat end mill
;TOOL/MILL,20,10,80,0 ;TOOL/MILL,1,.5,2 - ball mill (control point is the tool tip)
;TOOL/MILL,20,-10,80,0 ;TOOL/MILL,1,-.5,2 - ball mill (control point is the center of the ball)
;TOOL/MILL,20,3,80,0 ;TOOL/MILL,1,.125,1 - bull mill

The line format for Corner Rounding End Mill is described below:

;TOOL/CRMILL, Diameter1, Diameter2, Radius, Height, Length

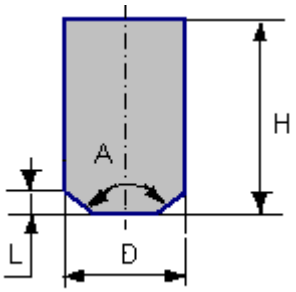


The line format for Chamfer tools is as follows:
;TOOL/CHAMFER, Diameter, Point Angle, Height

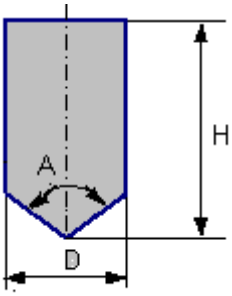


or

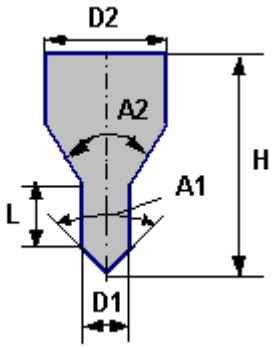
;TOOL/CHAMFER, Diameter, Point Angle, Height, Chamfer Length



The line format for Drill is described below:
;TOOL/DRILL, Diameter, Point Angle, Height



And the line format for Center Drills is as follows:
;TOOL/CDRILL, D1, A1, L, D2, A2, H (see Fig. below)



*Note: You can insert a new tool into the NC file at the needed position. To get the Tool definition dialog choose «Insert Tool definition» from the Edit window context menu. After a Tool line is inserted, you have to select the «Run»-»Rebuild» menu item or press Ctrl-F9 in order to implement the changes.
You can add a new tool to the tool database. Simply press the «Add tool to DB» button from the Tool definition dialog. Afterwards you can get the tool from this database pressing the «Get Tool from DB» button.

If you have the tool database then you can use the following syntax for the Tool definition command:

;TOOL/ID=……
or
;TOOL/ Description=……..

In this case CutViewer will use the tool from the database with given «ID» or «Description» value.

d) ;COLOR,R,G,B

You can assign display color to each tool. This must follow the TOOL
Command.

R - specifies the intensity of the red color
G - specifies the intensity of the green color
B - specifies the intensity of the blue color

The intensity for each argument is in the range 0 through 255.
If all three intensities are zero, the result is black.
If all three intensities are 255, the result is white.

For example, you may type the following:

;COLOR,0,255,0 - Display Color is green;
or
;COLOR,255,255,0 - Display Color is yellow;

Note: CutViewer inserts Color line automatically if you use Tool definition dialog. You can change Display Color for each tool pressing the color assignment button in «Setup» mode or directly using a simple key-combination «Ctrl-T». To change the color, simply click on the appropriate cell in the «Color» column.

Note: Any leading symbols are permitted, provided they do not interfere with program execution.

For example, you may type the following:

;FROM/120,0,0
or
$FROM/120,0,0

Stock / Tool selection in CLS and APT CLDATA (ASCII) files:

Each file must contain information about stock in following format:
PPRINT/BLOCK,d1,d2,d3,d4,d5,d6
or
PPRINT/CYLINDER,d1,d2,d3,d4,d5,d6

d1,d2,d3,d4,d5,d6 - see G code section

For example:
PPRINT/BLOCK,100.,70.,30.,50.,35.,30.

Information about tool for CLS files:

TLDATA/MILL,D,r.... = mill
TLDATA/DRILL..... = drill

Information about tools for CLDATA (ASCII) files:

CUTTER/D,r

D-tool diameter
r- corner radius
for flat end mill r=0
for ball mill r=D/2

To assign display color to each tool use command:
PPRINT/COLOR,R,G,B

Refer to the demo files for more examples.

Editing

You can modify an NC program in the Edit window, as you would edit a file. All changes will be accepted only after you select the «Run-Rebuild» menu item or press Ctrl-F9 for program rerunning.
For program debugging you can select the menu item «Run to Cursor» or simply press F4 key. The «Run to Cursor» command runs your program from the current point to the line the cursor is on in the edit window and then switches to the Step-by-Step mode. For example, if you are checking a large NC file you may select fast Turbo mode of operation then select the desired line in the edit window and press F4. When the program stops at selected line you can switch to the Solid mode and continue processing. You can also use Ctrl-F4 after modifying the program. This would be the same as «Rebuild» and «Run to Cursor».
You may also set breakpoints on lines in the NC file where you want the program execution to pause during a run. Running to a breakpoint is similar to Running to a cursor position in that the program runs until it reaches a certain line. However, unlike «Run To Cursor», you can have multiple breakpoints in your file. To set a breakpoint, select the line in the file where you want the breakpoint set, then do one of the following:

a) Press F5
b) Choose Toggle Breakpoint from the Run menu
c) Choose Toggle Breakpoint from the Edit window context menu.

When you set a breakpoint, the line on which the breakpoint is set becomes highlighted.

*Note: you can select Edit commands through popup menu. In order to open this menu the user should click with the mouse right button somewhere on the Edit window area.

Zooming

Use one from the following buttons for zooming:

Zoom Window

Allows for the zooming by a window or region selection method. To use this functions click the button and while holding down the left mouse button select the desired window and then release the mouse button.

*Note: The resulting part will be sectioned because of memory restrictions and for faster processing.

Zoom In

Press this button and then press the left mouse button at the desired point to Zoom in on a part. Press the right mouse button to leave Zoom In mode.

Zoom Out

Press this button and then press the left mouse button to Zoom out on a part. Press the right mouse button to leave Zoom Out mode.

Zoom Restore

Reverts to the initial zoom view.

View Rotation

For view rotation press the 'Rotate' button. To rotate the view, hold the left mouse button down while moving the mouse. When the view is in the desired position, press the right mouse button to leave «Rotation» mode.

*Note: you can enact the «Rotation» command directly using a simple hot key without pressing a button. Press the Alt key and click-drag with the left mouse button.

View Panning

For view panning press the «Pan» button. To pan the view, hold the left mouse button down while moving the mouse. When the view is in the desired position, press the right mouse button to leave «Pan» mode.

*Note: you can enact the «Panning» command directly using a simple hot key without pressing a button. Press the Ctrl key and click-drag with the left mouse button.

Sectioning

The part can be sectioned to see invisible elements. CutViewer supports two modes of sectioning, «Parallel to Axis» and «Through 2 points».

«Parallel to Axis» generates a sectioned part slicing it with a plane parallel to the Axis. Press the «Parallel to Axis» button and an outline of the top edges of the stock will appears. Place the mouse cursor on one of the lines and drag it to the location where you would like to see

the section cut. When you release the mouse an outline of the section will appear below the line. You may move the section line again with the mouse. Repeat this for other edges if necessary. When finally satisfied, press the right mouse button to create the section.

«Through 2 points section»- allows you to cut away a portion of the part through any vertical plane defined by two points. This option is available only on the Top and ISO views. Move the cursor to the first position and press the left mouse button down. While holding the button pull a «rubber band line» to the desired position. Release the mouse button and correct the position of the points as necessary. When you are satisfied with the position of the section line, move the cursor to the side of part you want to remove and press the right mouse button.

To restore the sectioned part press the «Unsectioning» button or choose the menu item «Section», «Unsectioning».

Measurements

A tool has also been supplied for making measurements of the part. For making measurements press the «Measure» button» and then press the appropriate button. The most of measurements are available on the Top view only.
To build sections through vertical planes press the «Measure»-»Section» button and input explicit coordinates of the first and second points and step value or move the cursor to the first position and press a left mouse button down. While holding the button pull a «rubber band line» to the desired position. Release the mouse button and correct the position of points as necessary. When you are satisfied with the position of the section line, press the right mouse button and input a step value in the dialog box. The section will appear on the screen. You can see real coordinates in the X,Y,Z box by clicking at a point of the section. You may zoom in on a section .You may also choose the «coordinates» page and save X,Y,Z values in the file.

*Note: The values of PART and REMOVED VOLUMES are approximate and relate to current display representation of the part (taking into account all sections).

STL Compare» function allows the user to perform a check of the cut model to 3D CAD model in STL format. For making comparison press the «STL Compare» button. The STL Compare dialog box appears. Load the STL file. You can see the following fields and the buttons:
dx,dy,dz and xa,ya,za are used for setting the STL orientation relatively to the program origin.
dx,dy,dz indicate the STL origin relatively to the program origin.
xa(ya,za) indicates the angle of STL rotation around the X(Y,Z)-axis. Press the «Show STL» button to show STL model and to make certain of STL position and orientation.
InTol - the tolerance value for the gouges.
OutTol - the tolerance value for the uncut areas.
Press the «Start» button to begin the comparison.
The process is slow, especially for big STL files. The gouges will be displayed in Red, uncut ares - in Yellow, and all area's within
tolerance - in Green colour. You can change the colours pressing the appropriate coloured buttons and choosing desired colour.

Settings

Through the «Setup» button you can change and save the following parameters:

Mode of Simulation
1. Solid: Graphically simulates the material removal process by updating the stock solid shape as the tool moves (by default).
2. Turbo: Graphically updates only the toolpath with the tool image (if «Show Toolpath» and «Show Tool» were activated). The machined part appears only after reaching the end of NC file or when the user presses «Step Forward», «Pause» or «Stop» button.

Autosave options
specify which options are saved automatically when CutViewer is closed. A check mark means it is enabled.
- Main window position: if checked then main window size and position are saved.
- GOW settings: if checked then settings for each graphic windows are saved.
Edit window position: setting position of the Edit window in the main program window.

Screen layout settings
specify the number of graphical windows(1,2 or 4), edit window position (left,right,top,bottom) and GOW orientation (Vertical or Horizontal).

Check Rapid Collision
enables or disables the detection of Rapid Collision. If this option is on then CutViewer graphically simulates any collision the tool has with the stock material while under Rapid Traverse. Execution of the NC file will stop at the block of code where the collision occurs and a special dialog box will also appear that will display the coordinates for the current type of move in a from-to format and the buttons to define next actions. You may continue execution, pause or stop it. If you press the «Ignore All» button then execution will continue but without any Rapid Collison detections or in other words the option «Check Rapid Collision» will be Off. That's why don't forget to turn «Check Rapid Collision» to On if you want to have this feature beeing available for further simulations in the same session.

Dwell (milliseconds)

specifies a dwell at the every screen point of tool motion for slowing the on screen tool speed

Trace step
specifies an amount of the tool motions for the Trace mode when the tool moves incrementally

Tool View
specifies Solid of Wireframe mode for the tool image

Colors
setting the color palette.

*Note: When CutViewer loads NC file, it creates a record in memory of any tool size or tool color statements that have been inserted where the tools take effect. A different color statement can optionally follow each tool definition.
If no color statements are found in the post, CutViewer uses the «Part color» shown in the Setup dialog. Changes made to tool colors by means of the setup dialog will persist only if there are no colors assigned in the NC file. A Ctrl-T key combination will directly launch the tool color dialog.
Reloading the NC file (Rebuild – Ctrl-F9) will cause the program to use any colors added to the editor buffer and override changes made in memory through the setup dialog. Once the NC post is loaded into memory the tool colors can be changed in the setup and seen immediately.
So, for a permanent color update we should add COLOR commands to the NC file.

Customizing

CutViewer uses a simple text file with an *.nci extension to configure CutViewer for specific style of post. The nci file uses the same format the common .ini file. These files are kept in the CutViewer install directory. The *.nci configuration file defines the extension filter and description that appears in the combo-box in the open file window. The extension filter can be used to limit viewing only those files for which the configuration file is correct.

The default configuration file called _gcode.nci may be found in the CutViewer directory. If you have only one NC machine simply edit this file. If you start by editing the default file it would be a good idea to save the original before editing.

Several sample *.nci files are provided with the program and can be used to view some demo posts. Move any move any of these sample nci files to another directory to shorten the list of choices in the file open combo-box.

If you have several machines, make copies of _gcode.nci under different names in the CutViewer directory and edit these files. Notepad or WordPad can be used to edit them. When saving these files, make sure the file is saved as FileName.nci and not as FileName.nci.txt (Notepad is notorious for this).

Note: A backup of the default _gcode.nci is also found under the name _gcode.def.

To load an NC file into CutViewer you must select the correct configuration file from the «Files of Type» combo-box in the file open dialog when you load a file.

Creating new NCI files

There are two required sections in this file:
[Common]
[Format]

NCI files are not case sensitive. Only the settings that are used by your program need to be in the file.

Comments have been added above each setting in a different font to show that they are not needed in the file.

A sample nci file looks like this:

[Comments]
A description that, when seen in the combo=box, will help describe what kind of post the configuration is for.
CNC=Fanuc OMC

The extension filter, which is used by the open file window to filter the file list, should be the same as what appears in you NC post filename.
FileExt=*.nc

Note: multiple extensions are allowed. Example: FileExt=*.nc;*.cnc

The character that the post uses before a line that tells the machine controller not to interpret that line is,
Comment_chr=(

The word Inch or Metric, depending on which unit scheme will be used in the NC post file.
Units=metric

[Format]

The next 3 settings define the coordinate symbols
x_addr=X
y_addr=Y
z_addr=Z

The next 3 settings define the ratio X,Y,Z value in NC file to actual value of X,Y,Z coordinates:

x_scale=1
y_scale=1

z_scale=1

The next two settings define how arc centers are read when either G90 or G91 is active and only apply to a G02/G03 line that has a center defined with I J K words but not when X Z R is used.

IJKformat_abs=
IJKformat_incr=

There are exactly 5 different string values that can be used with each:

I and J are absolute
Center.

I and J are incremental from the XY end point of the arc
end-center.

I and J are incremental from the start point of the arc
center-start.

If the control expects that incremental centers are always a positive number an «abs» is added.
abs(end-center)
abs(start-center)

If G90 active
Ijkformat_abs=

If G91 active
Ijkformat_incr=

Example:
IJkformat_abs=end-center
IJkformat_incr=abs(end-center)

The feed rate symbol
f_addr=F

The RPM symbol
s_addr=S

The tool symbol
tool_addr=T

The feed rate that above which rapid color is displayed and collision is expected if the cutter has met material.
f_rapid=5000

Cutter compensation g-codes.
cutcom_left=G41
cutcom_right=G42
cutcom_off=G40

Please note that if your CAM system is posting profiles using the cutter centerline instead of the part profile, each one of these should be set to «Gxx» so that they are ignored when encountered.

The end of the main program
prog_end=M30

Sub routine call

sub_call=M98P

Begin sub definition
sub_begin=O

End sub definition
sub_end=M99

Sub repeat character
sub_repeat=L

example of a _gcode.nci file :

```
[common]
CNC=Standard G-code
fileExt=*.nc
comment_chr=(
units=metric
[format]
x_scale=1
y_scale=1
z_scale=1
f_scale=1
s_scale=1
IJkformat_abs=center-start
IJkformat_incr=center-start
x_addr=X
y_addr=Y
z_addr=Z
f_addr=F
s_addr=S
tool_addr=T
f_rapid=5000
cutcom_left=G41
cutcom_right=G42
cutcom_off=G40
prog_end=M30
sub_call=M98P
sub_begin=O
sub_end=M99
sub_repeat=L
```

Custom Control Conversion

Some CNC have the canned cycles and special functions that can't be defined via variables in NCI-file. In this case the user should develop his own converter.

If there is the following line at [format] section in the nci-file:

user_convert=mycoverter.dll

then first of all CutViewer will try to call the procedure named User_convert_proc from myconverter.dll file. User_convert_proc procedure has
2 parameters - input file name and output file name. These parameters
CutViewer passes to your procedure. If you write your programs on Delphi the declaration for the procedure in DLL looks as follows:

Procedure user_convert_proc(infile,outfile:pchar);stdcall;

Input file is NC-file to be converted and output file is a result file with standard G-codes. In other words CutViewer calls the procedure Procedure user_convert_proc passing 2 parameters and expects that your procedure will create output file with the name passed by «outfile» parameter and then CutViewer handles this file as usual. There is one rule for output file. Every line in this file should be prefixed by number and «#» symbol. Example:

```
.....................
10# G01 X1 Y1
11# G00 Z5
14# T1
15# G01 X1 Y0.5 F2
.....................
```

Number before «#» symbol means that current line in the output file is corresponding to the line in input file with the index defined by this number (first line in the input file has the index 1) .

Let me explain.
We should convert NC-file for hypothetical CNC that has the following format:

```
TOOLNO, 1
SPINDLE, 500
RAPID
GOTO/X=1,Y=2,Z=5
FEEDRATE, 20
GOTO/X=1,Y=2,Z=1
FINI
```

This is the input file. Our procedure creates the appropriate output file with the following standard G-codes:

```
1# T1
4# G00 X1 Y2 Z5 S500
6# G01 Z1 F20
7# M02
```

You see that number before # is the appropriate line index for input file. CutViewer needs this index to provide correct cursor navigation in the edit window during step-by-step mode because the first file will be loaded into the edit window but in fact internally CutViewer works with the second file.

The algorithm how to convert input file into output file or in other words the program code inside the procedure is your responsibility and of course this file in details can be different for different developers. For example you can create the following output file in our case:

```
1# T1
2# S500
4# G00 X1 Y2 Z5
5# F20
6# G01 X1 Y2 Z1
7# M02
```

You can also create exe file instead of DLL and the line in NCI-file in this case is as follows:

```
user_convert=myconverter.exe
```

In this case at first CutViewer runs myconverter.exe file and passes 2 parameters through command line (input and output file names). Your program should accept these parameters and create the appropriate output file.

Contact points

For questions and orders please contact:

LAMSON GLOBAL CORPORATION
Web Site: http://www.cutviewer.com
Email Sales: sales@cutviewer.com
Email Support: support@cutviewer.com

About Register

The evaluation version of CutViewer only functions for 30 days.

To purchase CutViewer and deactivate the 30 day limitation contact...

LAMSON GLOBAL CORPORATION
Web Site: http://www.cutviewer.com
Email Sales: sales@cutviewer.com
Email Support: support@cutviewer.com