




Version 2.4.8
Revision date 13/08/2025



ART ToolShop



ART ToolShop



ART ToolShop



Version 2.4.8
Revision date 13/08/2025
























































ART Australasia Pty Ltd
57 Trade St Lytton 4178 QLD Australia
PH 61 7 3393 6555 FAX 61 7 3393 5355
www.advancedrobotic.com
sales@artcnc.com













































ART ToolShop






















Welcome.....	15
Installation instructions	16
Overview of menus	17
File menu	17
 New.....	17
 Open (Ctrl+O).....	18
 Save.....	18
 Save As (Ctrl+S).....	18
 Import (Ctrl+I)	18
 Export (Ctrl+E).....	19
 Export As Part	19
 Selection	19
 Sheet	19
 Undo (Ctrl+Z).....	19
 Output to Machine	19
 Exit.....	21
Recent File Options	21
View menu	21
 Select Material	22
 Add Sheet	22
 Delete Sheet	22
 Rotate Counter-Clockwise.....	23
 Rotate Clockwise.....	23
Zooming.....	23
 Zoom In	23
 Zoom Out.....	23
 Zoom Window.....	23
 Sheet	24
 Selected.....	24





	Extents.....	24
	Navigation.....	24
	Pan	24
	Orbit.....	24
	Standard Views.....	24
	Top View (F2).....	24
	Left (F3).....	24
	Front (F4).....	24
	Bottom (F5).....	24
	Back (F6).....	24
	Right (F7).....	25
	Diagonal SE (F8).....	25
	Diagonal SW (F9).....	25
	Diagonal NE (F10).....	25
	Diagonal NW (F11).....	25
	Show Cut Order.....	25
	Perspective.....	25
	Wireframe.....	25
Transform menu	26
	Select With Mouse.....	26
	Select by Layer (Ctrl+M).....	26
	Clear Selection	27
	Select All.....	27
	Move	27
	Linear Copy	29
	Array Copy.....	30
	Scale.....	30
	Mirror X.....	31
	Mirror Y	31














	Invert.....	31
	Reverse	31
	Rotate	32
	Rotate Clockwise 45 Degrees	34
	Rotate Anti-Clockwise 45 Degrees.....	34
	Delete Selected (Del)	34
	Combine (Ctrl+L)	34
	Cleanup Selected	35
	Break Apart (Ctrl+K)	35
	Group (Ctrl+G).....	35
	Smart Group (Ctrl+Y)	35
	Ungroup (Ctrl+U)	36
	Moving selected shapes to another layer	36
	Create Part	37
	Confirm Save Part options	38
	Create Part From Selection	39
	Edit Part.....	39
	Dissolve Part	40
	Show Part Names	40
	Drawing menu	41
	Create Shapes.....	41
	Modify Shapes	41
	Node Edit.....	41
	Node Edit Shape	41
	Node Edit Select.....	42
	Node Edit Move	43
	Node Edit Add	43
	Node Edit Subtract	44

	Node Edit Join	44
	Node Edit Break	44
	Node Edit Connect	44
	Node Edit To Arc	45
	Node Edit To Line	45
	Node Edit Chamfer	46
	Node Edit Fillet	46
	Draw Shape	46
	Create Shape	48
	Boolean Tools	50
	Offset Shape	51
	Text Tool	52
	Shapes	53
Toolpathing menu	54	
	Shape Check	54
	Shape Check Parameters	54
	Offset (Ctrl+T)	56
	Plasma Toolpathing Settings	57
	Router Toolpathing Settings	61
	Centre	68
	Fill	68
	Fill Settings	68
	Thread Milling	72
	Thread Milling Settings	72
	Marking	75
	Marking Settings	75
	Diamond (3D Engraving)	79
	Diamond Cut Settings	79
	Drill / Pierce	82
	Drill Toolpathing Settings	83
	Drill Point Placement Mode	85
	Continue Placing Drill Points	85
	Cancel	85
	Edit Drill Settings	85

	Print Head.....	86
	Edit.....	87
	Delete	87
	Multi Process	87
	All Templates Add new template.....	88
	All Templates Edit template	89
	All Templates Delete template	89
	Multi Process Template Add Multi Process Template.....	89
	Multi Process Template Delete Multi Process Template.....	90
	Link Cutting.....	90
	Continue Linking Toolpaths	91
	Cancel.....	91
	Remove Link.....	91
	Bridges	92
	Continue Editing Bridges	94
	Cancel.....	95
	Edit Bridge Settings	95
	Remove Bridge	95
	Edit Bridge	95
	Line of Bridges.....	95
	Edit Start Points.....	96
	Reset Start Points	97
	Top.....	97
	Bottom	97
	Left.....	97
	Right	97
	Top Left	97
	Top Right.....	98
	Bottom Left	98
	Bottom Right.....	98
	Layout menu.....	99
	Auto Nest.....	99
	Manual Nest	102
	Nest To Direction	103

	Nest To Left	103
	Nest To Right.....	103
	Nest To Bottom.....	103
	Nest To Top	103
	Add Rail-Cut	104
	Remove Rail-Cut	105
	Add Skeleton Cut.....	105
	Cut Order Manual	106
	Finish Cut Order	107
	Cancel.....	107
	Cut Order Columns.....	107
	Cut Order Rows.....	108
	Cut Order Tweak	108
	Cut Order Cut Order Settings	110
Printing menu	111	
	Nest Report Design	111
	Symbols	112
	Nest	112
{P}	Fields	112
	Nest Report Preview.....	113
	Nest Report Print	113
	Select Printer	114
	Select Printing Range	115
Options menu	116	
	Preferences	116
	Select Machine	124
	Post Processor Editor.....	125
	Edit Output Options	126
	Material library	127
	Layer Manager	128
	Tool Library.....	129

Adding New Tool.....	130
Deleting Tool.....	131
Adding Template.....	131
Selecting Template.....	132
Apply changes to this only filter buttons	133
OK button.....	134
Cancel button.....	134
 Tool Changer.....	134
Help menu.....	136
 About ToolShop.....	136
 Help.....	136
 Licences	137
Activating ToolShop	137
Keyboard Shortcuts	139
File Operations	139
Ctrl+O.....	139
Ctrl+S.....	139
Ctrl+I.....	139
Ctrl+E.....	139
Ctrl+Z.....	139
Standard Views	139
F2.....	139
F3.....	139
F4.....	139
F5.....	139
F6.....	139
F7.....	139
F8.....	139
F9.....	140
F10.....	140
F11.....	140
Keyboard Options for Selection.....	140
Modes:	140
List of keyboard options for selection:	140
Shift+Mouse click	140
Shift+Mouse marquee select	140
Ctrl+M	140
Ctrl+A.....	140
Keyboard Options for Editing	140
Modes:	141
List of keyboard options for editing:	141
Arrow keys	141
Shift+Arrow keys	141
Alt+Left Arrow key.....	141
Alt+Right Arrow key	142
Ctrl+Mouse drag part	142
Ctrl+Mouse rotate part	142
Angular Constraint Movement	142
Shift+Mouse drag part	142
Shift+Mouse rotate part	142
Ctrl+Shift+Mouse drag part.....	142

Ctrl+Shift+Mouse rotate part	142
Ctrl+D (Duplicate)	142
Del (Del)	142
Ctrl+L (Combine)	142
Ctrl+K (Break Apart)	143
Ctrl+G (Group)	143
Ctrl+Y (Smart Group)	143
Ctrl+U (Ungroup)	143
ATP Module	144
File Open (Ctrl+O)	145
File Import (Ctrl+I)	145
File Re-Open	146
Importing part files	148
Starting ATP session	149
Move to Layer	152
Printing	153
 Nest Report	153
 Part Labels Design	153
Set Label Layers	154
Symbols	154
 Shape Preview	154
 Part Compass	154
 Nest Indicator	154
{P} Fields	155
{P} Custom Property Fields	155
 Part Labels Preview	156
 Part Labels Print	157
 Nest Labels Design	157
Symbols	158
{P} Fields	158
 Nest Labels Preview	158
 Nest Labels Print	159
 Sheet Labels Design	159
Symbols	160
 Sheet Shape	160
{P} Fields	160
 Sheet Labels Preview	160



Sheet Labels | Print 161

Options | Preferences 161

Options | Material Library 162



Print Stock Labels for Selected..... 162

Material Templates 162



Options | Layer Manager 164

Producing Quality Drawings..... 166

Step by Step Guide to Cutting a Job..... 168

Step 1: DXF File..... 168

Step 2: Open ART ToolShop 168

Step 3: Select File | Open menu 168

Step 4: Select All..... 168

Step 5: Cleanup selected 168

Step 6: Combine 168

Step 7: Toolpathing..... 169

Step 8: Auto Nest..... 170

Step 9: Edit Start Points 170

Step 10: Cut Order..... 170

Step 11: Export CNC 170

ToolShop User Manual..... 171

1. Using List Files for Automatic Importing of Shapes 175

2. Using Nest Reports to Assist Machine Operators & Quoting 176

3. Reduce Human Error using Nest Labels 178

4. Track Stock with Sheet Labels..... 178

5. Part Labels..... 179

6. Using the designer 181

Welcome

Welcome to ART Australasia's Tool-Pathing software, ART Tool Shop version 2.4

ART ToolShop has been designed to provide all the functionality required to convert DXF format CAD files into usable CNC machine code files for use on the profile cutter. Although ToolShop provides a simple, easy to understand interface, you will find that the functions are very powerful and will enable you to perform most jobs with ease.

We hope you enjoy the simplicity of ART ToolShop. We understand that there are many more desired features that can be used in your industry, so we are working hard to create new functions. We welcome your input as to what features you would like to see added to the software in the future. All ART ToolShop customers will receive upgrades for the warranty period of their machine. This will give you access to any new processes that are developed during this time free of charge. Thanks again for choosing ART Toolshop.

Regards, The ART Software Development Team



ART Australasia Pty Ltd
57 Trade St Lytton 4178 QLD Australia
PH 61 7 3393 6555 FAX 61 7 3393 5355
www.artcnc.com
sales@artcnc.com

Installation instructions

Insert the CD-ROM into the drive on a PC running Windows 7 with Service Pack 1 or later. If the auto run feature does not activate, use Explorer to start the setup program on the disk.

Follow the prompts and select the default options. If your PC does not have some of the .NET Framework updates, these will be automatically installed at this time.

The setup program will automatically detect the best configuration. The 64-bit version of the ToolShop is installed by default if the PC environment allows it. Otherwise, 32-bit version of the ToolShop is installed. The setup program will also add a ToolShop icon to the START menu.

ART ToolShop is designed to be compatible with most CAD drawing software that can export in DXF format.

If you have any difficulty in transferring files, first see the chapter entitled "Producing quality drawings".

If you have any continuing issues, please contact the ART help line on 61 7 3393 6555 (ask for software support) or call your local distributor for help. Alternatively you can contact us by email support@artcnc.com



Overview of menus

The purpose of this chapter is to introduce the basic methods and functions of ToolShop.

File menu

To access the File menu click the round "TS" icon on top left corner of the main menu.

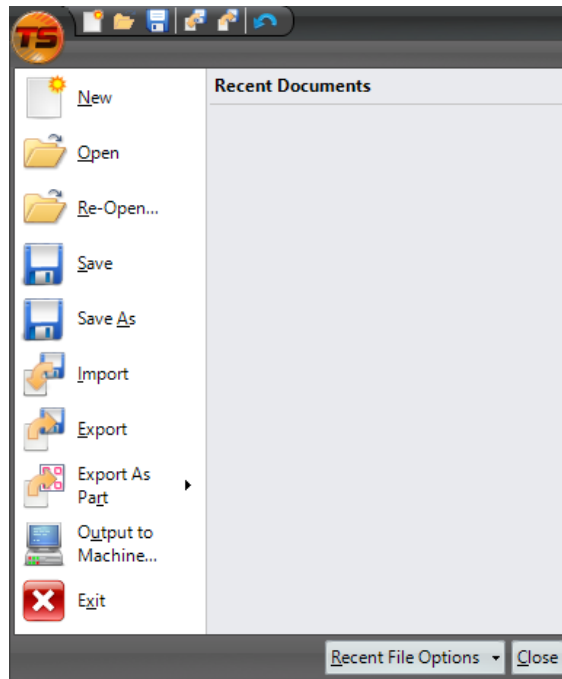


Figure: The file menu allows you to perform standard procedures similar to most Windows based applications



Figure: Some of the file operations are also available on this quick launch toolbar. The associated icons are shown beside the menu item.



New

The New function is designed to clear the entire job from memory and start a new job. Before doing so you'll be prompted to make sure that you wish to start a new job. If you click "Yes" then ToolShop will ask you to select the correct material and dimensions. Please see the section "View | Select Material".



Open (Ctrl+O)

This function will access the file open dialog box. Use this window to navigate to and open the DXF files that have been previously exported from your CAD program. Please see the chapter “Producing quality drawings” for more information on the format and style of drawings required for processing.

In addition you can open ToolShop job files (*.TSNEST). Previous ToolShop versions saved job files as (*.ATS) files. You may still open and save jobs in ATS format. However, TSNEST format is recommended as it supports a larger number of drawing shapes, loads faster and produces a much smaller file size.

You can also open ToolShop part files (*.TSPART). These files can contain a single part, or a job sheet exported as a part.

Note: Some additional features are available with the optional Automatic Tool Path (ATP) module. For more information please see the chapter titled “ATP Module”



Save

This function will access the file save dialog box. Use this window to save your current job into a ToolShop job file. Job files have *.TSNEST file extension. This function is only available if you have opened a drawing or you have imported shapes into ToolShop work area.

Note: Please note the information under “Open” command regarding *.ATS format.”



Save As (Ctrl+S)

This function will access the file save as dialog box. You can use this window to save your current opened job file under a different filename. This function is only available if you have opened a drawing or you have imported shapes into ToolShop work area.



Import (Ctrl+I)

Add new parts to an existing job. If all of the parts for the current job are not included in one CAD file, use this function to bring in more parts to add to the current job. Imported part will be automatically selected.

You can also import ToolShop job files. However when you import a job file (ATS or TSNEST), the material and worksheet information will be ignored. This means that all the shapes in the file will be imported to the current worksheet. The part files (TSPART) can only be imported to material that matches the part material.

Importing CNC files is possible as well as reviewing the actual cut path, cut order and some of the toolpathing parameters. After importing a CNC file choose “View | Show Cut Order” from the main menu to show the cut order. Some of the toolpathing parameters can be viewed by clicking the “Toolpathing | Edit” menu.

Note: Some additional features, such as importing multiple files simultaneously, are available with the optional Automatic Toolpath (ATP) module. For more information please see the chapter titled “ATP Module”

Commented [JR1]: Added.



Export (Ctrl+E)

Save a file for use by other applications. Use this function is to create CNC files for use by your ART profile cutter. You also have the option to export DXF files to be read by other CAD programs.

Note: See also the next topic **Output to Machine**.



Export As Part

The ToolShop part is like a grouped selection. The part can contain sub-groups, shapes, toolpaths, internal cut order, bridges etc. Using parts can be beneficial for some complicated shapes where there is a need for several toolpaths and the processing order of those toolpaths is critical. The parts are saved into physical files with the extension (*.TSPART).

Note: See also the topic **Transform | Create Part**



Selection

Export selection as a part file. The function is also available if you click on the selection and choose "Export Selection As Part" from the pop-up menu.



Sheet

Export currently active work sheet as a part file. The function is also available if you click on the currently active sheet and choose "Export Sheet As Part" from the pop-up menu.



Undo (Ctrl+Z)

With the help of Undo function you can reverse any changes. However changes to the view, opening a new file or creating a new project cannot be reversed.



Figure: The Undo operation is available on the quick launch toolbar.



Output to Machine

This is same as the "File | Export" function with CNC file type as output. This function opens a dialog box which enables you to export the toolpath to a NC file.

Clicking Save button when Save as type is "*.nc; *.cnc" will show an additional dialog which confirms the action from the user. You may export just the selected shapes by selecting "Selection". If several sheets are defined, export "All sheets" can be selected.

The "Override cut order" option can be used to define the order in which tools are used or shapes are cut.

The "Force All First" option forces selected toolpath types to be cut first. You can alter the "Force All First" processing order, if necessary, e.g. you have more than one option selected.

For example, you may wish to do the marking or drilling for the whole sheet first before proceeding cutting the individual part. After the “Force All First” toolpaths are processed the remaining toolpaths are processed using the usual cut order override settings.

Commented [JR2]: Added

Commented [JR3]: Added

Use “Up” and “Down” buttons to change your cut order preference, the tool, toolpath type and layer order. Only the Cut order preference items that appear in **bold** text have effect on the cut ordering. The higher the “Cut Ordering” item is on the list the less other preferences can affect ordering. The “Specific layers” option allows user to export only selected layers.

The options "Show All Layers", "Show Layers in This Job Only" and "Show Layers on This Sheet Only" will allow you to filter the 'Layer order' list.

Clicking Cancel will return the Export dialog box where the directory or the file name can be altered.

The “Printing” options can be used to print labels or the nest report immediately after export.

Note: Please see the section “An Introduction to Reporting and Labeling in ART ToolShop” for more information on these options.

- **Print part labels on export.** Part labels are printed for every 'group' of shapes in your ToolShop nest on specific layers. These labels can allow you to track the individual destination of each part, from a DXF through to the final cut part.
- **Print nest labels on export.** This option prints a nest label detailing the filename, tooling, material and other details of the job. Nest label sticker may be printed and applied to sheets of material that are waiting to be cut.
- **Print nest report on export.** This option prints a graphical job report to assist machine operators and quoting.
- **Print remnant sheet labels on export.** This option prints a stock label if there is a newly created remnant.
- **Add remnants to library.** This option adds new remnants to the Material Library.

If the “Show print preview before printing” option is selected then a preview window is shown for each selected print option. The “Preview file on export” will open the just created CNC file in a text editing program.

Note: The “Override cut order” and “Output Specific layers” options can be also accessed from the “Options | Preferences” menu.

Commented [JR4]: Pic updated.

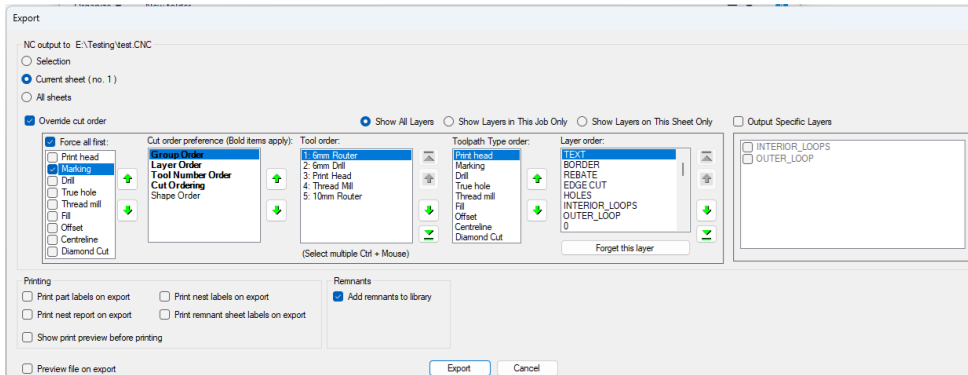


Figure: Exporting the currently selected toolpathed shapes to a machine file. User is asked to confirm the action.

Note: Output to Machine will export empty sheets as empty NC files.



Exit

Close the application.

Recent File Options

The options are as follows:

Clear Recent Files

This will clear the recent file history.

Set Number of Recent Files

This will open the "Input Recent Item Count" dialog box which allows you to limit the number of items in the recent file history.

Close

Closes the File drop-down menu.

Note: Some additional features are available with the optional Automatic Tool Path (ATP) module. For more information please see the chapter titled "ATP Module".

View menu



Figure: The View menu allows control over the display.



Select Material

Specify the dimensions and the material of the current sheet. Select from one of the predefined materials. Then select appropriate thickness. The Default Size drop down shows available standard sheet sizes for the selected material thickness. Use the Width and Height text entry boxes to customize the sheet size. If you define a custom sheet size then the sheet will be added as a remnant. To add to the predefined materials click the "Material Library" button.

Note: Toolshop will prompt user to select material on program start-up.

Figure: The Select material allows the user to specify the dimensions and the material of the current sheet.

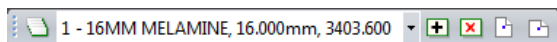


Figure: Change material for the current sheet by clicking "Select material" button on this toolbar.



Add Sheet

You can use this function to add new sheets to your job. This function will show same window as in "Select Material".



Delete Sheet

Use this function to remove sheets from your job. If you remove a sheet that contains objects the programs asks you whether you want to move the objects to a new sheet, delete objects or move them to another sheet. The "Add Sheet" button allows you to define a new sheet for the objects.

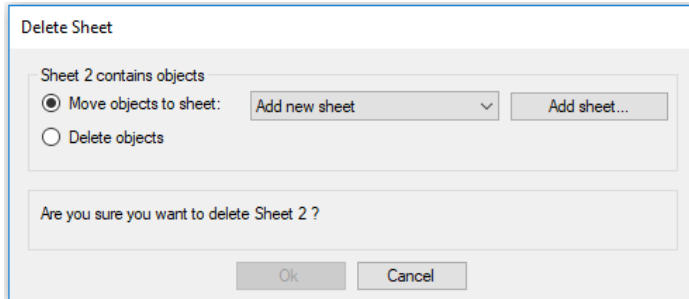


Figure: The Delete Sheet allows the user to delete any sheets and specify how contained objects are handled.



Rotate Counter-Clockwise

Use this function to rotate whole sheet counter-clockwise in 90 degree increments.



Rotate Clockwise

Use this function to rotate whole sheet clockwise in 90 degree increments.

Zooming

The following options are used to zoom in or zoom out from the job.



Figure: The same functions can also be accessed through this toolbar



Zoom In

The display zooms into the centre point of the screen. If you use the mouse wheel to zoom in then it uses the mouse cursor as the focus point.



Zoom Out

The display zooms out from the centre point of the screen. If you use the mouse wheel to zoom out then it uses the mouse cursor as the focus point.

Note: You can reverse the mouse wheel zooming direction from the "Options | Preferences | General settings | Inverse wheel zoom direction".



Zoom Window

Allows you to select a rectangular area to which to zoom in.



Sheet

Zooms into the current worksheet.



Selected

Zooms into the selected shapes.



Extents

Zooms to the extents of the job.

Navigation

These options allow you to move the view sideways or orbit it around the job.



Pan

Allows you to move the current view sideways.



Orbit

Allows you to orbit the current view around a central point. You can also access this function by holding the middle mouse button down and then dragging the mouse.

Standard Views

The following options are used to position your viewport of the job. Activating one of these functions will position the viewport at the desired location and centre the job within the screen.



Top View (F2)

This is the standard ToolShop view.



Left (F3)

This is the view from the left.



Front (F4)

This is the view from the front.



Bottom (F5)

This is the view from the bottom.

Back (F6)

This is the view from the back. (Not in the ribbon bar but can be accessed via keyboard.)

Right (F7)

This is the view from the right. (Not in the ribbon bar but can be accessed via keyboard.)

**Diagonal SE (F8)**

This is a diagonal view from the South East.

**Diagonal SW (F9)**

This is a diagonal view from the South West.

**Diagonal NE (F10)**

This is a diagonal view from the North East.

**Diagonal NW (F11)**

This is a diagonal view from the North West.

**Show Cut Order**

Display the cut sequence number near the start point for each toolpath. This is the order in which the shapes will be processed on the machine. Cut order numbers are only visible for toolpathed objects.

**Perspective**

Toggle between isometric and perspective views. With perspective selected, an object further away appears smaller, while objects closer to the viewpoint appear larger. This is particularly useful when using a diagonal view. If perspective is not selected, all objects appear the same size irrespective of distance from the viewpoint.

**Wireframe**

Toggle between wireframe and rendered views. With wireframe selected, all 3D objects are rendered without material.

Transform menu



Commented [JR5]: Pic updated smart grp, show part names

Figure: The transform menu allows you to perform Selection operations and to modify selected objects.



Figure: The Selection operations are also available on the above toolbar.



Select With Mouse

Select using the mouse. Drag a box around specific objects for selection. Alternatively use the mouse to click on a single object. Click outside of the selected object on an empty area to deselect.

To use the point and click method is to select multiple objects, hold down the *Shift* key while clicking an object. This will add objects to the selection without clearing any previous selections. Click on a part already selected while holding down the *Shift* key the selection will be reversed for that object.

Note: When selecting objects on the ToolShop screen, the marquee selection only works on shapes that are completely visible. For example the holes might be on a different layer than the outside shape. However you can select the shapes that are only partly visible by left clicking the visible section of the part. Please see the section "Layer Manager" for more details on how to hide individual layers.



Select by Layer (Ctrl+M)

With the help of the Select by Layer function you can select objects on a specific layer. Tick the layers you want to include in the selection. This is useful when CAD drawings are separated into different layers for different types of objects. ie: Holes, Boundary cuts, Fold Lines etc.

Note: Use this function to apply a specific toolpath to entire layer of objects or delete unwanted objects on a layer such as annotations etc.

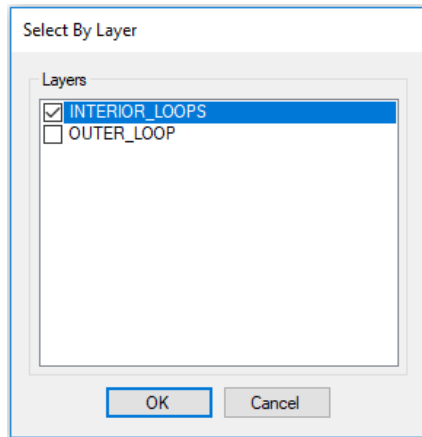


Figure: The Select by Layer allows user to select objects on specific layers.



Clear Selection

Clear the selection rectangle and deselect all parts on the screen.



Select All

Highlight all parts on the screen and display the selection rectangle around them. This is useful for selecting all parts that have been brought in from a cad drawing for further processing.

Note: When activating a function that modifies parts on the screen that function will usually only apply to selected parts.



Move

Enables selected objects to be moved accurately.

Select move by offset or by absolute X,Y position. The Move function does not change the Z coordinates of the objects. If the Absolute mode is used, a reference point needs to be specified. Choose the selection rectangle reference point to be moved to the absolute position. When a reference point is selected the X, Y fields show its current coordinates. If Absolute mode is selected you can also select whether to include toolpaths and lead-ins/lead-outs in the reference point calculation.

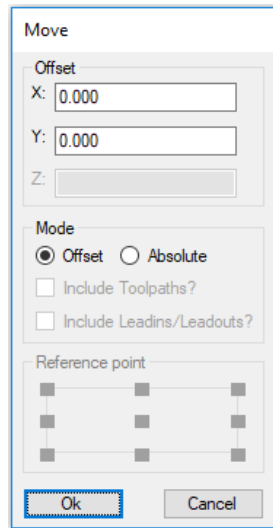


Figure: Move selection by offset.

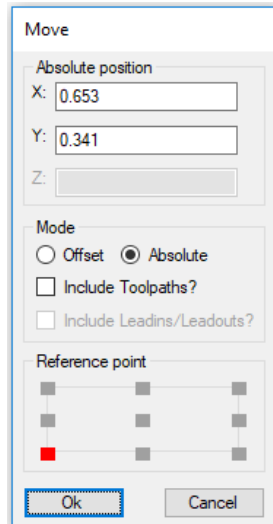


Figure: Move selection to an absolute position

Note: You can also manually move the selected objects on the screen by using the mouse. After selecting objects, you will see the selection box appear along with the yellow move node in the centre.

Click and hold the left mouse button on the yellow move node.

By dragging with the left mouse button down you will be able to move the selection rectangle.

During the dragging the mouse cursor is displayed as a moving cursor.

Releasing the left mouse button completes the move process.

Cancel the function anytime by clicking right mouse button down and selecting Cancel Move from the pop-up menu.

You can also switch between Move function and Copy function while you are dragging the selection by pressing and holding the Control key on your keyboard. The original objects will stay where they were and a copy will appear at the new position.

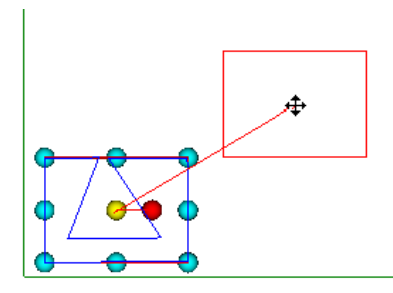


Figure: Moving selection by mouse.

Note: If you try to Move a shape that is on a locked layer a small padlock sign is shown above the yellow centre node.



Linear Copy

Make several copies of the selected objects. Specify the offsets in X and/or Y directions. In the Quantity field specify the number of copies to be made.

Linear Copy

Offset

X: 10

Y: 0.000

Z:

Quantity

1

Ok Cancel

Figure: Making one copy of the selected items with X offset = 10.

Note: Objects can be manually copied by selected objects on the screen by using the mouse. After selecting objects you will see the selection box appear along with the yellow move node in the centre.

Press and hold the Control key on your keyboard and then Click and hold the left mouse button on the yellow move node.

By dragging with the left mouse button down you will be able to move the selection rectangle. During the dragging the mouse cursor is displayed as a moving cursor with a small + symbol beside it.

Releasing the left mouse button completes the copy process.

Cancel the function anytime by clicking right mouse button down and selecting Cancel from the pop-up menu.

You can also switch between Copy function and Move function while you are dragging the selection by releasing the Control key on your keyboard. The objects will then be moved to the new position.

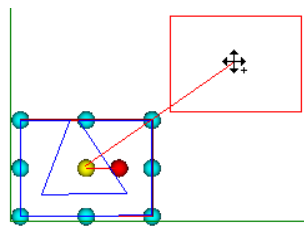


Figure: Making a copy out of the selection by using Control key and the mouse.



Array Copy

Make an array of the selected objects. This is a very handy function to fill the whole work sheet with copies of a specific imported item. Specify the offsets or gap in X and/or Y directions. Then define how many columns and rows required. For example, defining 2 columns and 2 rows makes 3 more copies out of the original so that the total number of objects, including the original, is 4. The Array copy function will create items in the order shown in the below figure, which is optimal for cutting purposes.

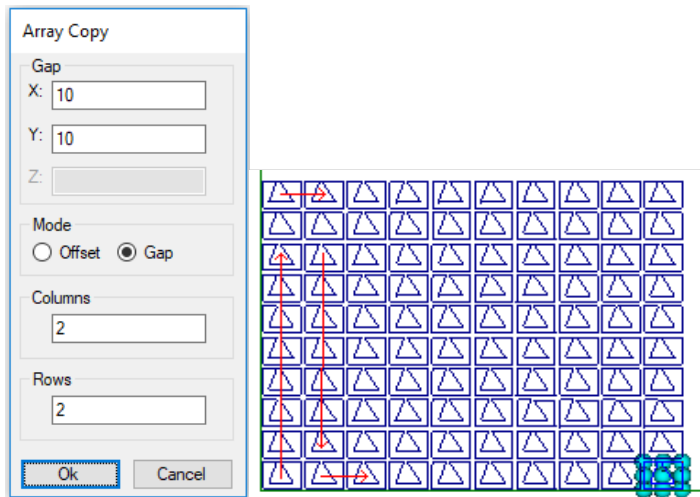


Figure: Making an 2 x 2 array of the selected items with X, Y offsets = 10. Array function will create items in order which is optimal for cutting.



Scale

This function allows you to scale the selected shapes. Specify scale values in X and/or Y directions. After selecting the scaling unit, choose the selection rectangle reference point from which the scaling is calculated. If you switch from the Percent unit to Millimetres/Inches it will show you the new size of the selection in that unit and vice-versa.

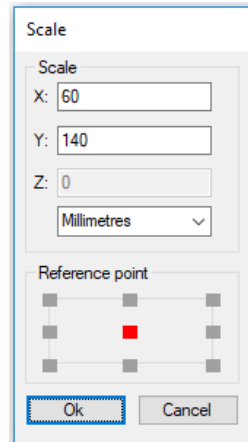


Figure: Scaling selected shapes.



Mirror X

This will flip the X coordinates.



Mirror Y

This will flip the Y coordinates.



Invert

Normally, an internal shape such as a hole will automatically be linked to an outside shape using the Combine function. This forces the toolpath to be applied to the inside of the hole.

However, if you wish to manually change the toolpath side from outside to inside (or vice-versa) you may use this function. Simply select any shape(s) and apply. You will see the colour change to denote the gender.

This allows different settings to be applied to an inside shape, or allows cutting inside a shape that has no outside parent shape.

If you wish to keep inverted shapes linked to external shapes, then you should use the group function rather than the Combine function. This will not override the Invert function.



Reverse

Reverses the direction of all open contours in the current selection.

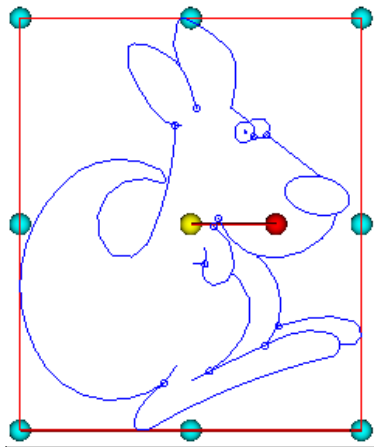


Figure: A drawing made mostly from open-contours. Open contour start points are indicated by little circles.



Rotate

Rotate the selected objects accurately by coordinates. Select the rotation point either as an offset from the centre or as an absolute position. Offset X = 0 and Y = 0 rotates object around its centre. Specify the angle as degrees. A positive angle value will rotate in anti-clockwise direction. Apply duplicate function is practical for creating circular arrays of objects. Clicking the button will make a copy of the object, makes it active selection and then rotates it. Pressing the button continuously creates a circular array of objects.

Rotate

Absolute position

X:

Y:

Z:

Mode

☐ Offset
 ☒ Absolute

Angle

Apply duplicate

Ok

Cancel

Figure: Rotate selection anti-clockwise 45 degrees around its centre.

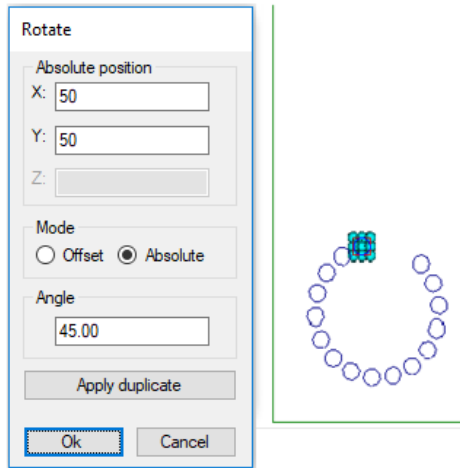


Figure: Creating an array of circles around the point X = 50, Y = 50 with 20 degrees steps by using the Apply duplicate button.

When the Absolute mode is selected, the X, Y fields shows the current centre coordinate of the active selection.

Note: You can manually rotate the selection on the screen.

After selecting objects you will see the selection box appear along with the yellow move node and the red rotate node.

Click and hold the left mouse button on the red rotation node.

By dragging with the left mouse button down you will be able to rotate the selection rectangle.

During the dragging the mouse cursor is displayed as a rotation cursor.

Releasing the left mouse button completes the rotation process.

Cancel the function anytime by clicking right mouse button down and selecting Cancel Rotate from the pop-up menu.

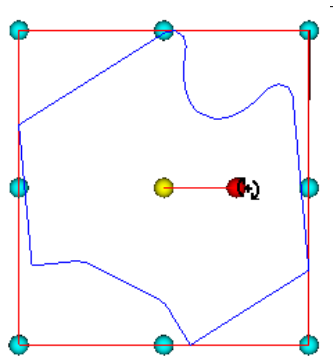


Figure: Rotating selection by mouse.



Rotate Clockwise 45 Degrees

This function rotates the selection 45 degrees clockwise.



Rotate Anti-Clockwise 45 Degrees

This function rotates the selection 45 degrees anti-clockwise.



Delete Selected (Del)

Delete all selected objects on the screen. You can also delete the selection by pressing the Delete key from the keyboard.



Combine (Ctrl+L)

If your drawing includes parts with internal holes, this function will automatically separate the parts into external and internal shapes. This is vital for several reasons. Firstly, holes must be cut on the inside of the line while outside shapes are required to be cut outside. One other reason for this function is to define the direction of cut. The plasma cutting process requires holes to be cut in an anti-clockwise fashion. External paths must be cut in a clockwise direction.

For the Combine to process the holes, the holes must be on the same layer as the outside shape. If the holes are required to be on a different layer, you may use the "Invert" function on the hole shapes to reverse the direction of the cut.

Commented [JR6]: Combine function change in behaviour.

This function can also be performed automatically when a part is imported. To change this option, please go to the "Options | Preferences menu". Click on the "Import Settings" tab at the top of the window. Tick the "Combine On Import" box on or off depending on your preferences.

Note: This function was called "Create Hierarchy" in the previous versions of ToolShop.

Note: This function is also affected by the optional ATP module. For more information please see the chapter titled "ATP Module"

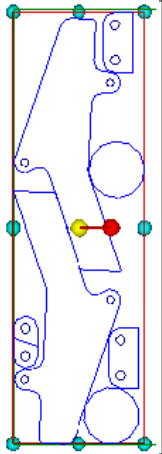


Figure: Drawing with no combined shapes. Internal shapes are drawn with red colour.

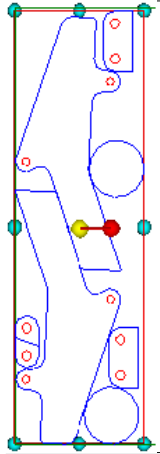


Figure: Drawing after Combine operation. Internal shapes are drawn with red colour.



Cleanup Selected

This function will join up open corners which are usually present in DXF drawings. Some cad programs will explode the drawing into multiple line segments. These are unsuitable for machining. Using cleanup will restore the drawing to a usable state.

Note: ToolShop does not perform cleanup on text entities.



Break Apart (Ctrl+K)

Breaks apart combined shapes and returns the shapes to individual paths. This allows separate functions to be performed to each path individually. This may be used if you desire to apply different tool path parameters to internal and external shapes.



Group (Ctrl+G)

Groups selected objects into one entity.



Smart Group (Ctrl+Y)

Smart Group is a quick way of linking all objects contained within the perimeter of a parent object so it can be nested as a single part.

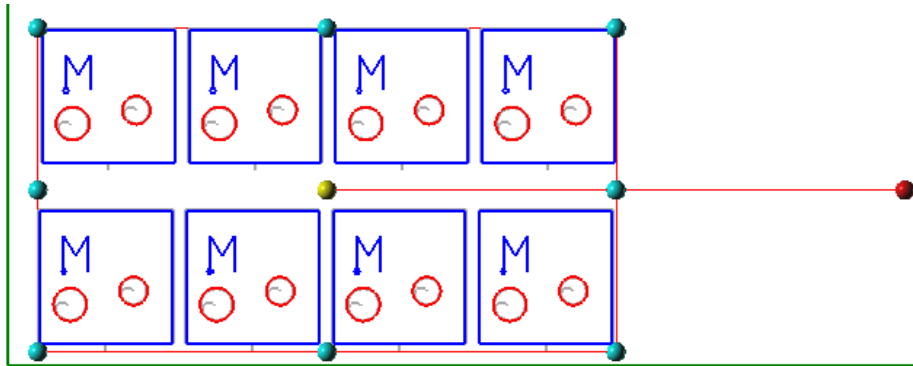


Figure: Executing “Smart Group” in above scenario creates 8 individual parts (groups). The traditional “Group” function would create a single part.

Commented [JR7]: Smart Group added.



Ungroup (Ctrl+U)

Ungroup the selected groups into individual entities.



Moving selected shapes to another layer

To access the "Move to Layer" dialog box right click on the selection and choose “Move to Layer” from the pop-up menu or you can use the “Layers” drop-down list box to achieve this.

When you click on a shape the “Layers” list box shows you the current layer the parent shape is on.

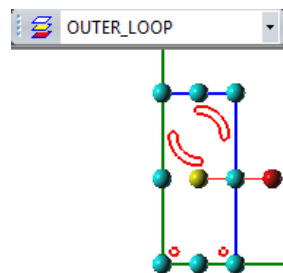


Figure: In this example the holes are on “INTERIOR_LOOPS” layer and the outer shape is on “OUTER_LOOPS” layer. When shape is selected the “Layers” list box shows the parent layer.

However, if you select multiple shapes that are on different layers the “Layers” list box shows “[Multiple Layers]”. You can proceed to change the layer for the whole selection by simply selecting another layer name from the list box. User is then shown the "Move to Layer" dialog box to OK or cancel the change. The layer change will only affect the parent shapes.

See the section “Layer Manager” for more details on how to add additional layers to the job. These additional layers will be then shown in the “Move entities to Layer” list box.

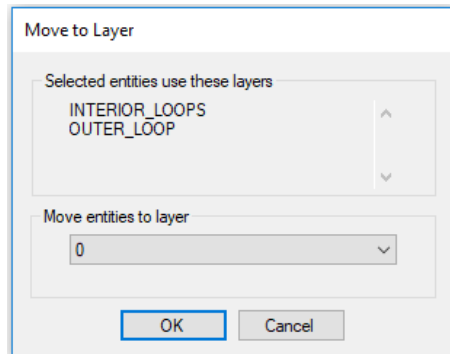


Figure: Use Move to Layer dialog box to move selected objects to another layer.



Create Part

Clicking "Create Part" menu will open a blank part work area. This allows you to create new parts from the scratch. Within the part work area, you can import CAD shapes into the part, edit the shapes and perform toolpathing and cut order functions.

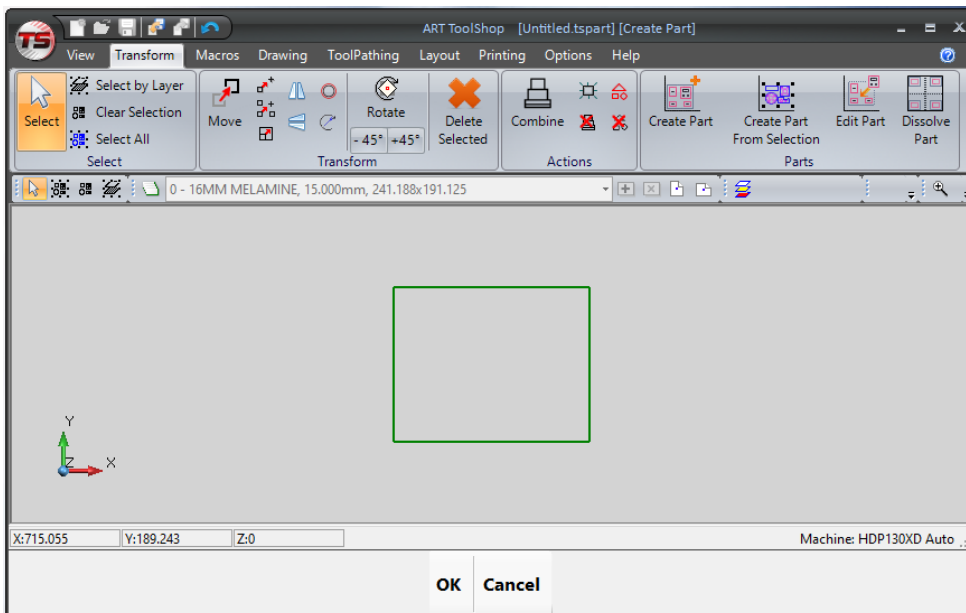


Figure: ToolShop with a blank part work area.

The part work area is painted with a grey background colour. You can change this colour from "Options | Preferences | Colours | Part Editing Background". The material drop down list shows the default part material. By default, this is the same than current job material.

Use File | Import function to import CAD drawings (*.DXF) into the part work area. Alternatively, you can use the ToolShop CAD program by clicking “Drawing | Create Shapes” or you can choose one of the “Drawing | Tools” toolbar options to create shapes. ToolShop operations that are not allowed within part work area will appear greyed out.

If you wish to discard any part modifications, click the Cancel button. This will exit the part work area and will return ToolShop to the normal operation.

Click OK button to save the part. You will be then prompted to choose a filename for the part. To accept the filename, click the Save button.

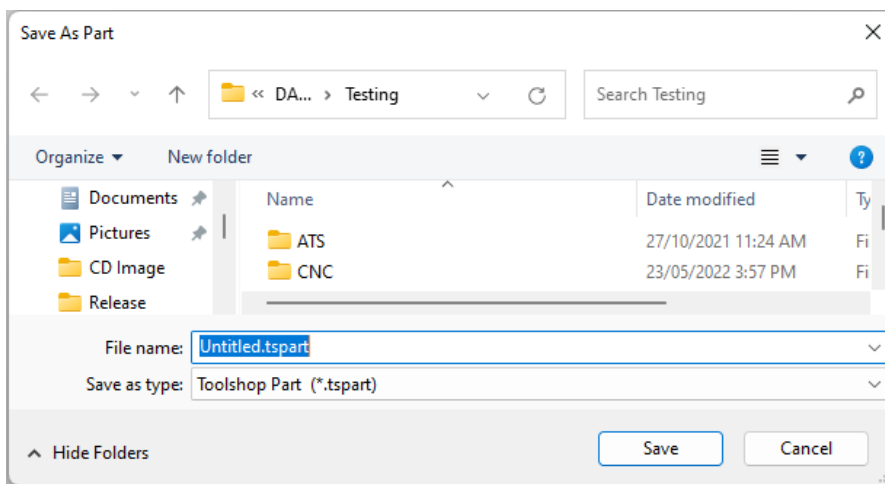


Figure: Choose filename for the part.

Clicking the “Save” button will then open the “Confirm Save Part” window. Options that are not allowed are greyed out.

Confirm Save Part options

- **Save.** This is the default option when editing a part. The modifications to the part are saved into the original part file. The “Update all copies of this part in the current job” is automatically selected. This means that all copies of the part in the current job will be updated as well.
- **Save As.** You can save the modified part with a different filename. This allows you to keep the original part file untouched. The “Update all copies” option is selected by default. This means that all copies of the part in the current job will be replaced with this modified part. Unselect the “Update all copies” option to save the changes to the new part file only.

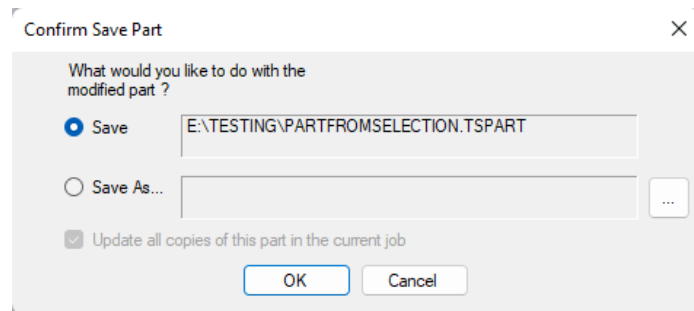


Figure: Confirm Save Part window.



Create Part From Selection

Using the mouse, select the shapes on the active work sheet you wish to form into a single part. Then click the “Create Part From Selection” menu. The selected shapes will now appear on the part work area. Then add any modifications, toolpaths or cut ordering to the part. When a part is created from a selection of shapes, the original individual shapes are grouped together.



Edit Part

To modify the shapes and the toolpaths that form the part, select the part by using the mouse and then click “Edit Part”. This will open the part in the part work area. The name of the edited part filename will appear in the ToolShop caption bar. The material that was selected during part creation is shown in the material drop down list. You can alter the material by clicking “Select Material” button. This material information is used by the ATP process.

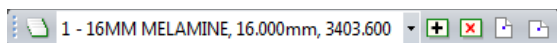


Figure: Change the material for the part by clicking “Select material” button on this toolbar.

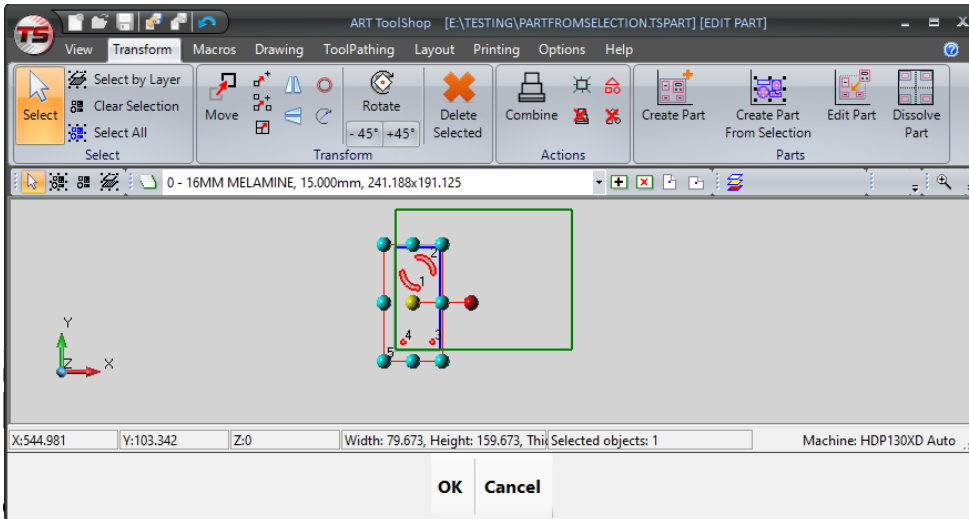


Figure: Editing a part.

Click OK to save any modifications to the part.



Dissolve Part

To break the selected part into its original components, select the part by using the mouse and then click "Dissolve part". The operation removes the part from the current job and the selection is replaced with its individual components. This operation has no effect on the physical part file.

Show Part Names

By default, this option is selected. It will show the individual part names.

Commented [JR8]: added



Drawing menu

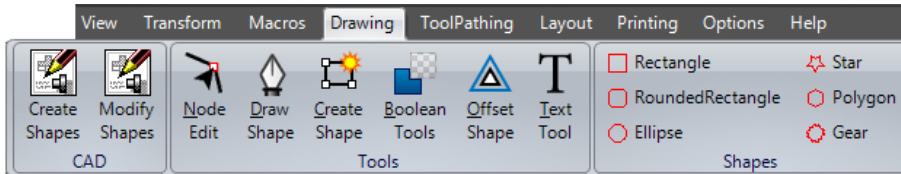


Figure: The drawing menu allows you to modify selected objects and to perform drawing operations.



Create Shapes

This function will allow you to create new shapes by using integrated ART Cad module.

Note: This additional feature is only available with the optional ART Cad module. For more information please see the chapter titled “ART Cad module”



Modify Shapes

This function allows user to modify selected shapes by using integrated ART Cad module.

Note: This additional feature is only available with the optional ART Cad module. For more information please see the chapter titled “ART Cad module”



Node Edit

This function opens the “Draw Tools” window which allows you to move, add, subtract, join, break and connect individual nodes of the selected shape. In addition drawing segments can be changed into arcs or lines. The “Draw Tools” window also enables you to add chamfers and fillets.

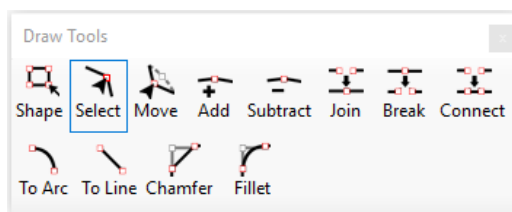


Figure: The “Draw Tools” window allows you to edit the individual nodes of the selected shape.



Node Edit | Shape

This function will highlight the nodes of the entire selected shape. Select the “Shape” option on the Draw Tools window. Then using the mouse, click on the shape which nodes you wish to highlight. You can also highlight the nodes of a shape by simply clicking it with the mouse.

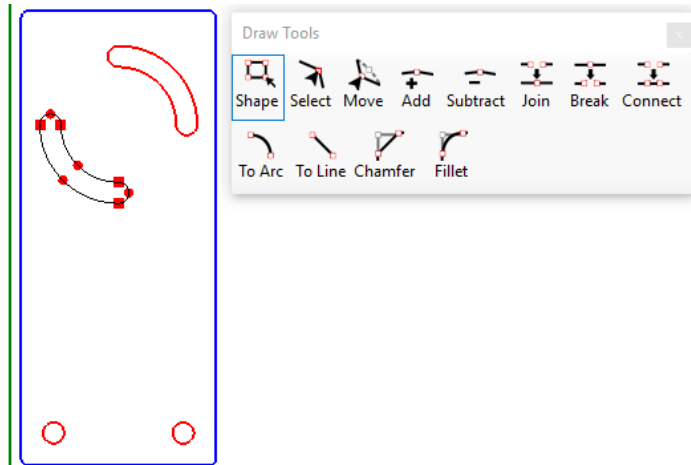


Figure: User has used the “Shape” option to highlight the nodes of a shape.

Note: When a shape is selected for node editing, combined shapes are automatically broken apart. Exiting the “Node Edit” will automatically re-apply the Combine function and any existing toolpaths.



Node Edit | Select

This will allow you to select the nodes you wish to edit. Select first a shape by clicking with the mouse. The nodes for that shape will be highlighted. The end nodes are shown as red squares and mid points as red circles. Use the mouse to click on a node you wish to edit. Small “x” or reference point indicator is shown for the first selected end node. Alternatively you can drag a box around specific nodes for selection. Click outside of the selected node on an empty area to deselect.

To use the point and click method to select multiple nodes, hold down the *Shift* key while clicking a node. This will add nodes to the selection without clearing any previous selections. Click on a node already selected while holding down the *Shift* key the selection will be reversed for that node.

When you select a node the “Draw Tools” window expands to show the “Move Selected Nodes” option. If you wish to move a single node, proceed by selecting “Absolute” and it will show you the current X, Y coordinates of the node. Then input the new X, Y coordinate and click the “Move Selected Nodes” button. However, if you want to move a selection of nodes, then you may wish to use “Relative” option with X, Y offset values. This will move each selected node by an offset.

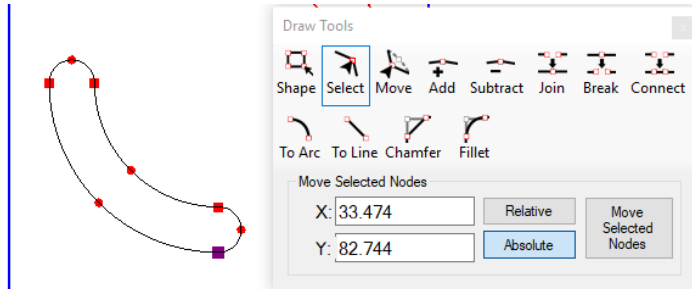


Figure: User has selected an individual node for editing.



Node Edit | Move

Click this to activate the “Move” mode. The mouse cursor changes into “Move” cursor. Press left mouse button down and drag the selected node or nodes into desired location. You can also move the selected nodes directly by simply pressing left mouse button down above the nodes and then dragging them. The mouse cursor changes into “Move” cursor whenever it is above the selected nodes.

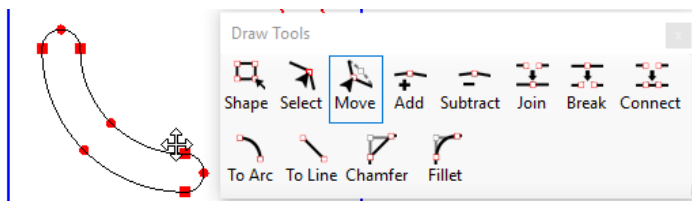


Figure: User is moving the individual node by using the mouse.



Node Edit | Add

Click this to activate the “Add” mode. In this mode the mouse cursor changes into “Add” cursor if it is above the segments of the selected shape. Click left mouse button at desired location to add the new node. A red square is added to distinguish the new node. While you are in the “Add” mode a stop sign cursor is shown where a node cannot be added.

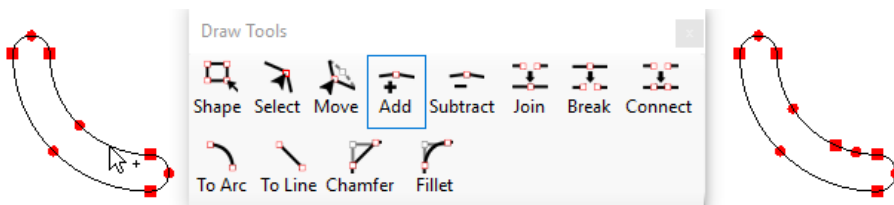


Figure: User has added a new node.

Node Edit | Subtract

This function will ask confirmation whether selected nodes will be removed.

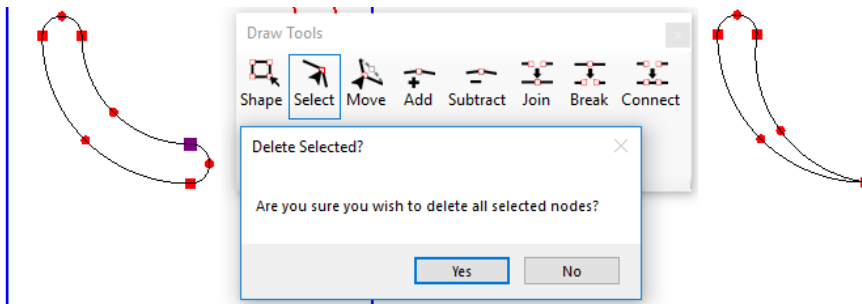


Figure: User has clicked "Subtract" to delete selected nodes.

Node Edit | Join

This option can be used to bring the two end nodes of the shape together making an open contour shape into a closed shape. Select first the two end nodes and then Click the "Join" button. Unlike "Connect" function which adds a new line, the "Join" function does not add a new line but simply moves the end points together.

Node Edit | Break

This option can be used to split a closed shape into an open contour. Also you can split an open contour in to two parts. However open contour cannot be split from the end points.

Node Edit | Connect

This option can be used to add a new line to connect the two selected end nodes of the same entity.

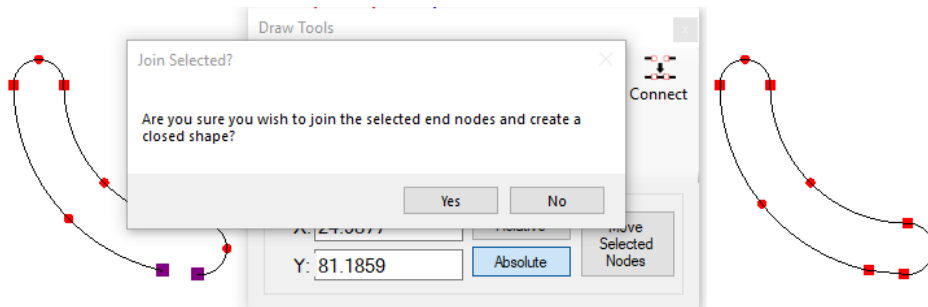


Figure: User is using "Connect" function to connect the selected end nodes with an extra line.



Node Edit | To Arc

Please use the “Select” function to select the shape you wish to modify. Then click on the “To Arc” function. If this mode is activated you can click on a single line segment to change it to an arc. After this the function returns to the “Select” mode. However, if you wish to change multiple line segments to arcs then hold down the *Shift* key while clicking on a line segment. A special cursor is shown for segments that can be changed into arcs. “To Arc” function adds a small red circle to the middle of a line segment to indicate that it is now an arc. You can drag the small circle to change the radius of the arc. When the “To Arc” option is not available a stop sign mouse cursor is shown.

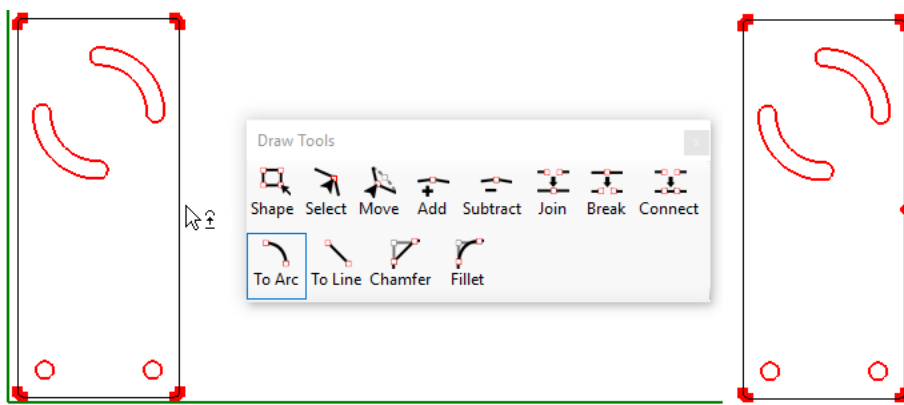


Figure: User is changing a line segment into an arc using “To Arc” function.



Node Edit | To Line

The “To Line” function works in a similar manner as the “To Arc” function. It will allow you to change arc segments into lines. After selecting the shape with “Select” function, click on the “To Line” function. Then click on an arc segment you wish to change into a line. After this the function returns to the “Select” mode. To change multiple arcs to lines, hold down the *Shift* key while clicking an arc segment. When the “To Line” option is not available a stop sign mouse cursor is shown.

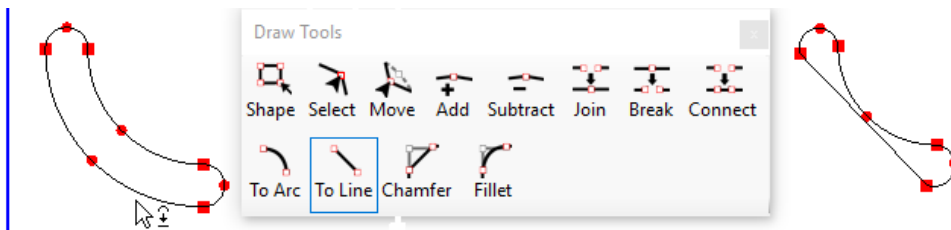


Figure: User has changed an arc segment into a line using “To Line” function.



Node Edit | Chamfer

Use this function to clip sharp corners. Proceed by first selecting the nodes ("sharp corners") and then click on the "Chamfer" function. Input the chamfer radius and then click "OK".

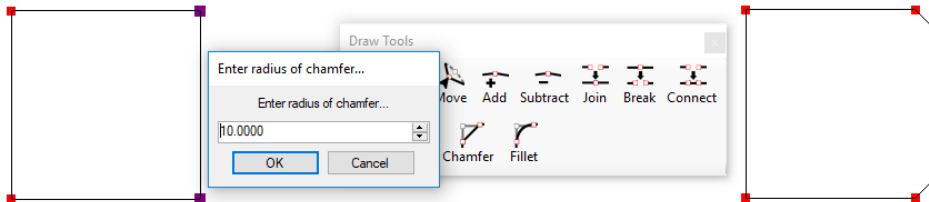


Figure: After selecting 2 nodes user has executed the "Chamfer" function using the radius of 10.



Node Edit | Fillet

This function can be used to add a basic fillet into a sharp corner, e.g. the corner is changed into a round corner. First select the nodes you wish to add the fillet to. Input the fillet radius and then click "OK".

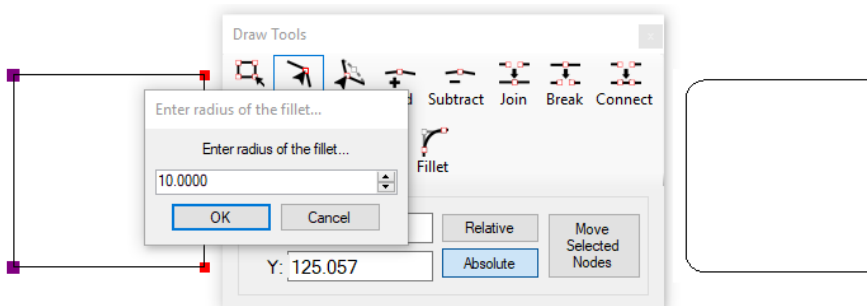


Figure: After selecting 2 nodes user has executed the "Fillet" function using the radius of 10.



Draw Shape

This function opens the "Drawing" window which allows you to draw a new shape into your job. The shape can be made up of line, arc or spline drawing entities. As the default setting the "Line" option with X, Y and Z location coordinates is active.

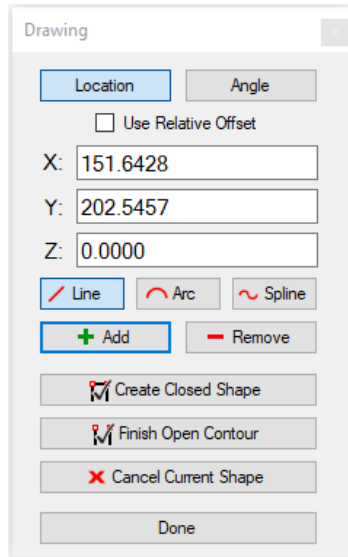


Figure: The “Drawing” window allows you to draw new shapes.

If you wish to start your shape creation with a line, then simply click on the screen where you want the line to start. Then drag the mouse cursor to desired end point and click left mouse button down. Alternatively you can input the exact X, Y, Z coordinate for the start point. After inputting the X, Y and Z values, click “Add” button to add the new start point. A start point node is drawn on the screen. You can input the exact values for the end point as well or drag the mouse cursor to the correct end coordinate. Then click “Add” button and the new drawing entity is drawn on the screen. If you don’t know the exact coordinates, you can use the “Angle” option. When this option is selected you can specify distance and angle from the previous point.

When a new drawing entity is added to the job its end point is used as the starting point for the next entity. However, if you do not wish the next line, arc or spline to start from previous end point, then click “Finish Open Contour”. Then select a new starting point for the next drawing entity. When you are finished with the complete shape then select “Done”.

The “Drawing” options are as follows:

- **Location.** Input X, Y, Z coordinates for the start and/or end point of the drawing entity.
- **Angle.** Input Deg. (‘angle’), Dist. (‘distance’) and Z for the start and/or end point of the drawing entity.
- **Use Relative Angle.** If this option is ON then the new angle value is relative to the previous position.

Supported Drawing Entities:

- **Line | Arc | Spline**
- **Add.** Click this option to add start or end point at user inputted coordinates.

- **Remove.** Click this option to remove previously added start or end point. Click multiple times to remove several start and end points.
- **Create Closed Shape.** You can click on this option anytime to make currently drawn open contour shape into a closed shape.
- **Finish Open Contour.** As default the starting point of the new drawing entity is the end point of the previous drawing entity. Clicking this option will finish the current open contour shape. This means that you can choose a different starting point for the next drawing entity.
- **Cancel Current Shape.** This will only cancel the current drawing entity. If you clicked "Create Closed Shape" or "Finish Open Contour" after adding your previous drawing entity, then these entities are not cancelled.
- **Done.** This will exit the "Draw Shape" function. User is asked confirmation whether to keep the shape. Select "Yes" if you wish to keep the newly created shape. If the newly created shape is an open contour then user is also asked "Do you wish to join this shape?" Select "Yes" if you wish to make the new shape a closed shape and "No" if you wish to leave it as an open contour.



Create Shape

This function opens the "Generate Shape" window which allows you to create a single shape or several shapes from a fixed list of basic shapes. The shapes that are supported are: Rectangle, Rounded Rectangle, Ellipse, Star, Polygon and Gear. The default shape is Rectangle. After selecting the appropriate shape you can create the shape in two ways:

- 1) Using the mouse. Press left mouse button down on the screen and then hold it down while dragging the mouse. A resizable rectangle is shown on the screen. This is the size of the new shape. After releasing the left mouse button the dimensions and other properties are shown on the "Generate Shape" dialog box. To alter the dimensions, enter the new dimensions into the fields or use mouse to drag from one of rectangle corner nodes. To change the location, you can enter the new coordinates or you can drag the shape with mouse from the central node.
- 2) Using keyboard. Select the appropriate shape and then click "New Shape" button. Then customize the shape by entering values into the configurable fields. A live preview of the shape is drawn on the screen at the specified coordinates. Click "Finish Shape" to save the shape.

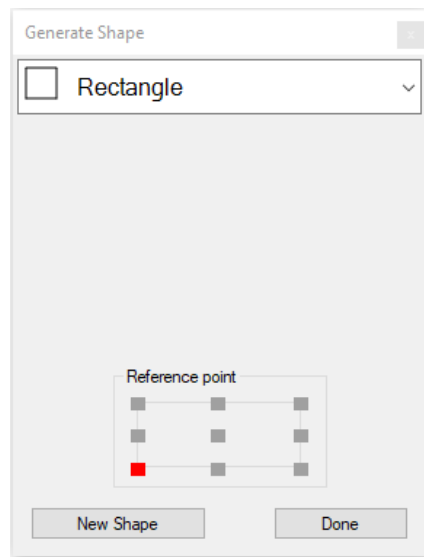


Figure: The “Generate Shape” window allows you to create basic shapes.

The “Generate Shape” options are as follows:

- **Supported shapes.** Rectangle, Rounded Rectangle, Ellipse, Star, Polygon and Gear.
- **Reference point.** This is used in positioning the shape into X, Y and Z coordinates.
- **Finish Shape.** Generates the current shape and then allows you to create another one.
- **Done.** Generates the current shape and then exits the “Create Shape” function.

Configurable fields for the basic shapes:

- **Rectangle.**
 - Width
 - Height
 - X
 - Y
 - Z
- **Rounded Rectangle.**
 - Matching Corners. If this option is ON then all the corners have the same radius.
 - ON
 - Corner Radius
 - Width
 - Height
 - X, Y, Z
 - OFF
 - Top Left Radius
 - Top Right Radius
 - Bottom Left Radius

- Bottom Right Radius
 - Width
 - Height
 - X, Y, Z
- **Ellipse.**
 - Width
 - Height
 - Maintain Ratio. If this option is ON then the Height value is ignored.
 - X, Y, Z
- **Star.**
 - Diameter
 - Inner Diameter
 - Number Of Points
 - X, Y, Z
- **Polygon.**
 - Sides
 - Width
 - Height
 - X, Y, Z
- **Gear.**
 - Diameter
 - Number Of Teeth
 - Tip Percentage
 - Tooth Height
 - X, Y, Z

Select the appropriate shape and then customize it by entering values into the configurable fields. A live preview of the shape is drawn on the screen at the specified coordinates. Click "Done" to close the



Boolean Tools

This function opens the "Boolean operations" window which allows you to perform Boolean operations between two selected shapes. The supported Boolean operations are: Union, Subtract, Intersect and Difference. Select two overlapping shapes and then choose one of the Boolean options. Performing a Boolean operation will modify the original shapes. However, if you wish to keep the original shape A or shape B or both of them, then set "Keep A" and "Keep B" options accordingly. A live preview of the end result is highlighted on the screen.

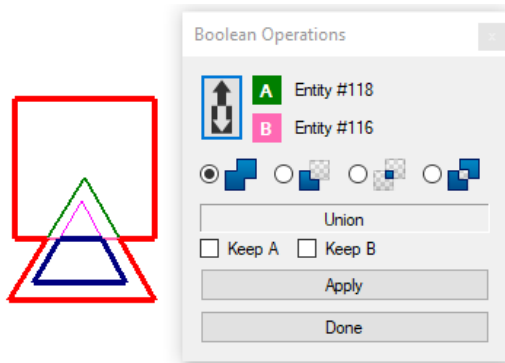


Figure: The “Boolean Operations” window allows you to perform Boolean operations between selected shapes.

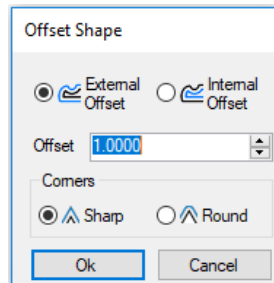
The “Boolean Operations” options are as follows:

- **Swap entity from A to B and vice versa.** This is the order the shapes are processed in. This affects the end result of the “Subtract” option. You can also swap A and B entities by mouse clicking on the edge of the selected shapes.
- **Boolean operation.** The supported Boolean operations are:
 - Union. Merging two shapes into one.
 - Subtract. Subtraction of one shape from another.
 - Intersect. Portion common to both shapes.
 - Difference. Portion not common to both shapes.
- **Keep A.** Keep the original shape A.
- **Keep B.** Keep the original shape B.
- **Apply.** Performs the selected Boolean operation and then allows you to select other shapes for Boolean operation.
- **Done.** Performs the selected Boolean operation and then exits the “Boolean Tools” function.



Offset Shape

This function opens the “Offset Shape” window which allows you to create a smaller or larger copy of the selected shapes. It offsets a parallel shape by a set distance. The new shape is created on top of the selected shape.



The “Offset Shape” window allows you to create a different size copy of the selected shapes.

The “Offset Shape” options are as follows:

- **External Offset.** The offset path is created on the outside of the shape.
- **Internal Offset.** The offset path is created in the inside of the shape.
- **Offset.** Define the offset distance.

Available corner options:

- **Sharp**
- **Round**

T Text Tool

This function opens the “Text Tool” window which enables you to create or edit text shapes. The text shapes can be further processed for print head or cutting. Select the font and style and input the text height. Then select one of the Text Alignment points by clicking with the mouse. The alignment point is used in positioning the text with mouse.

Then select the location for the text shape by clicking once on the screen where you want to create the text shape. A blinking caret appears on the screen. Then type in desired text. You may still change any of the text properties. Press “Enter” button to finalize the text shape. Alternatively, you can click anywhere on the screen and user is prompted to commit the changes.

You cannot move the text shapes to other locations until you have exited the “Text Tool” function. However, if you wish to create another text shape, simply select the appropriate text properties, click once with mouse on desired coordinates and type in the new text. To avoid editing the existing shapes click further away from them.

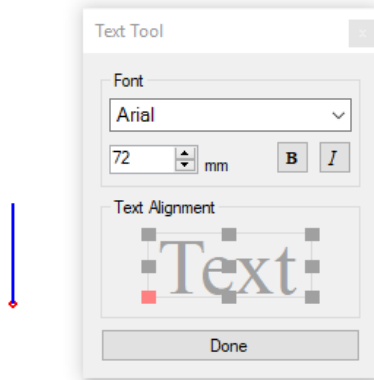


Figure: User is adding a new text shape into the job.

If you wish to edit any of the existing text shapes simply click on the near vicinity of that shape. Commit any changes by pressing “Enter” on the keyboard or by selecting another text shape with mouse. Alternatively, you can select a text shape you wish to edit before choosing the “Text Tool” function.

Note: Selecting a text shape that already has a toolpath for editing will cause the toolpath for that shape to be deleted. You will need to re-toolpath the text shape after editing.

Click “Done” to save any changes and to exit the “Text Tool” function.

The “Text Tool” options are as follows:

- **Font.** Select the font for the text shape.
- **Height.** Input the height of the text.
- **Style.** Bold or Italic.
- **Text Alignment.** The alignment point is used to position the text shape on the screen.
- **Done.** Exits the “Text Tool” function and also saves the current text shape.

Shapes

The Shapes menu gives you a quick access to “Create Shape” options. Click on one of the supported shape types: Rectangle, Rounded Rectangle, Ellipse, Star, Polygon and Gear to create that shape. The “Create Shapes” dialog box is then opened where you can alter the properties. Please see the “Create Shapes” section for more details.

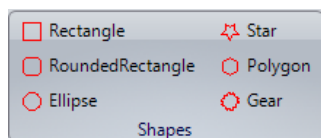


Figure: Shapes menu gives quick access to “Create Shapes” options.



Toolpathing menu

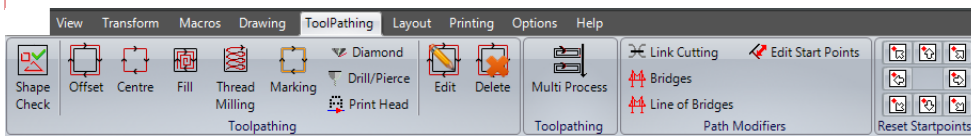


Figure: The Toolpathing menu allows user to do machining related operations.

Commented [JR9]: Pic updated. Marking & line of bridges added



Shape Check

To use "Shape Check" function first select shapes on the screen. These must have been previously processed using the "Cleanup" and "Combine" functions for this function to work correctly. By Supplying appropriate parameters through the Choose Shape Check Parameters dialog box, a check will be performed for possible problem areas when toolpathing.

Shape Check Parameters

Tool Name or Template name
The name of the current tool or template

Cutter Diameter or Cutter Size
The diameter of the cutter

Check Open Contours
Select Yes or No.

Open Contour Offset Direction
Centre: The toolpath will follow the path of the open-contour.
Left: The toolpath will provide the cutting path to the left of the open-contour
Right: The toolpath will provide the cutting path to the right of the open-contour.

Maximum tolerance for 'co-linear' lines
Default value is 1.000.

Choose what you would like to check for:

Unreachable segments
Segments that cannot be physically reached by the cutting tool. E.g. internal hole is too small for the cutter etc. Not applicable to "Diamond Fill" or to centerlining toolpaths such as knife, pen or printer tools. Default value is No.

Crossovers
Check for places where the path crosses over itself. Default value is Yes.

Co-linear segments
Check for places where the path doubles back on itself. Default value is Yes.

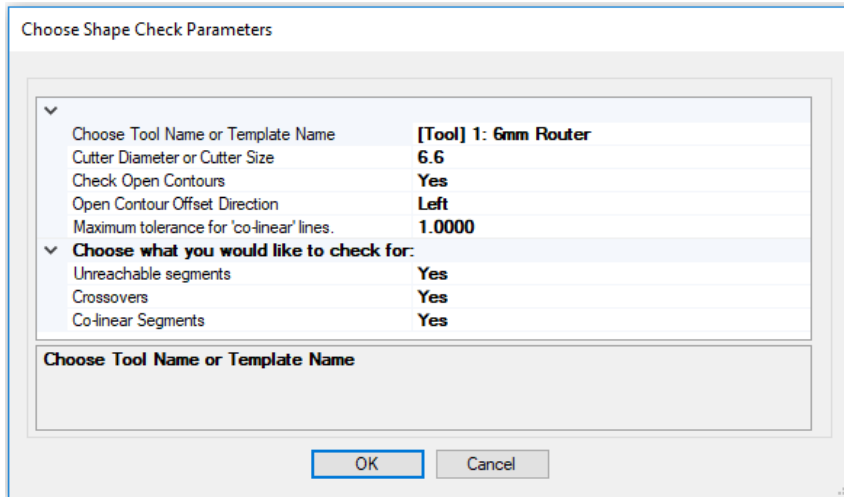


Figure: An example of shape check parameters.

If "Shape Check" function does not find any problem areas it will show "No potential problem areas were found" message box. However, if potential problem areas were found, it will open the "Potential Problem Areas" dialog box. A close-up picture of a particular problem area is shown on the left-hand side "Problem area" panel along with a brief description. The "Shape" panel shows the shape in question including a red rectangle identifying the problem area. The "Sheet" panel shows the location of the shape in relation to the sheet.

By using the Left and Right arrow buttons you can browse through the problem areas. Clicking "Ignore" button will remove current problem area from the list. When there are no more problem areas in the list the dialog box is automatically closed. Clicking "Ignore All" will close the dialog box straight away. Click the "Edit" button if you wish to manually edit the line segments in the problem area. ToolShop will then close the "Potential Problem Areas" dialog box and will activate the "Drawing Tools" for node editing.

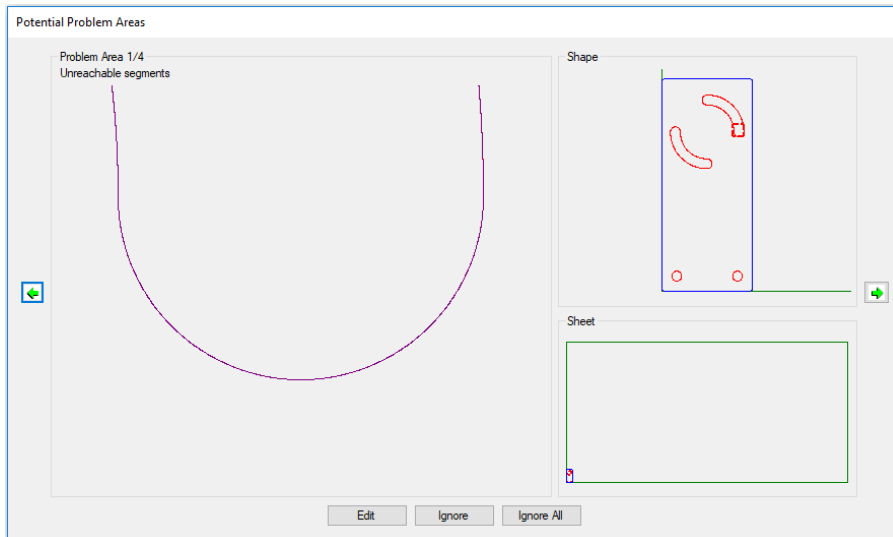


Figure: ToolShop has identified 4 toolpathing problem areas. In this example the cutter is too wide to cut the internal holes.



Offset (Ctrl+T)

To use "Offset" toolpathing function first select closed shapes on the screen. These must have been previously processed using the "Cleanup" and "Combine" functions for this function to work correctly. By supplying appropriate machining parameters through the "Choose Offset Toolpathing Parameters" dialog box, a toolpath will be generated. This must be done before the nesting and output to machine stages. The user can also save the toolpathing parameters as a template.

Note: The specific offset toolpathing options depend on the current machine configuration.

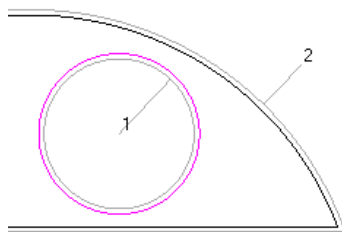


Figure: Section of an offset-toolpathed shape with 10mm straight lead-in in one of its holes.

Note: You can add additional toolpaths to the selected objects by repeating the "Offset" Toolpath function. The program asks first users acceptance. You will see this same message

also if your current selection already contains some shapes with a toolpath and you are adding a new toolpath.

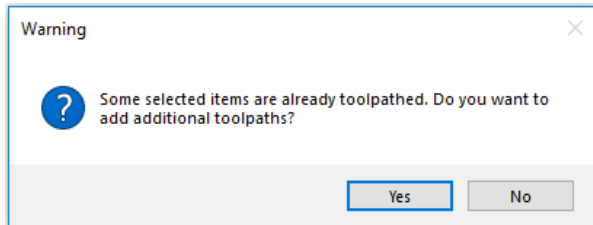


Figure: By repeating the "Offset" toolpath function you can add additional toolpaths.

Plasma Toolpathing Settings

1. Cutting Settings

Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Plasma Torch

Cut Width

The width of the cut.

Cut Height

The clearance between the torch and the material during the cutting process.

Pierce Delay

The time taken before the tool moves, after piercing the material.

XY Feed Rate

The XY feed rate.

Pierce Height

The clearance between the torch and the material during the piercing process.

Over burn / Underburn

This setting can be used to configure how cutting is finished for individual shapes. Define over burn by entering a positive value in and underburn by entering a negative value. The unit is mm. Default value is zero.

Has Sharp Corners?

Specify whether sharp corners are used in toolpathing. Default setting is Yes.

Reverse Direction

Default value is false.

TrueHole Spiral %

Default value is zero.

2. Open Path Toolpathing

Apply toolpath to open contours?

Select Yes or No.

Offset Direction

Centre: The toolpath will follow the path of the open-contour.

Left: The toolpath will provide the cutting path to the left of the open-contour.

Right: The toolpath will provide the cutting path to the right of the open-contour.

Colinear Leadin

Performs a reverse-direction leadin along the eventual toolpath. For short but deep toolpaths, it may need to zig-zag downwards. Not recommended for plasmas. Default value is false.

3. Lead-In / Lead-Out

Lead-in Type

Select Straight, Arc, FollowTheSide or None.

Lead-in Length

The length of the lead-in.

Lead-in Angle

The angle of the lead-in.

Lead-out Type

Default value is None.

4. Pre-Piercing

Perform Pre-Piercing?

Specify whether pre-piercing is performed.

Pre-Piercing Tool

The pre-piercing tool name.

Type

Drill or Plasma Torch (the pre-piercing settings depend on the selected tool)

The pre-piercing settings shown below are for a drill.

(Drill) Depth

The distance to drill.

(ActualDepthPerPass) This value is calculated automatically.

(DepthOfFirstPass) This value is calculated automatically.

(MaximumDepthPerPass) This value is calculated automatically.

Relative Depth

Input the depth relative to the "RelativeLocation" value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom

Below Surface

Above Bottom

(Drill) Z Feed Rate

Plunge speed.

(Drill) Dwell

The time taken before the tool moves, after piercing the material.

(Drill) Drill RPM

Drill revolutions per minute.

(Drill) Pre-piercing Offset

The distance between the pre-piercing point and the start of the main tool cut. This is ignored when a smaller drill is used to pre-pierce for a larger drill.

(Drill) Number of Pecks

Specify the number of drill pecks.

(Drill) Peck Retract Distance

Distance the drill retracts after each peck. This is an offset value from tool's current depth.

Note: If you wish the 'Peck Retract Distance' to be an absolute value from the material surface then set the option '**Retract above surface**' to True in "Options | Preferences | Toolpathing Settings".

Pre-piercing Settings for a plasma torch:

(Plasma Torch) Cut Height

The distance of the torch to the material to use whilst piercing.

(Plasma Torch) Pierce Delay

The time taken before the tool moves, after piercing the material.

(Plasma Torch) Pierce Height

The clearance between the torch and the material during the piercing process.

(Plasma Torch) AVC Volts

Plasma power setting for pre-piercing.

(Plasma Torch) Pre-piercing Offset
Pre-piercing offset in mm.

5. Multiple-Pass Settings (**This setting is not usually used with Plasma cutting.**)

Use a Clean-up Pass?

Specify whether clean-up pass is performed.

Clean-up Pass Thickness

Primary cut is offset this amount from the shape. The Cleanup pass removes this material and polishes the edge of the part.

Remove Bridges?

Yes or No.

6. Miscellaneous Settings

Place Bridges at Regular Intervals?

Specify whether bridges will be automatically added.

Bridge Repeat Distance

The distance from the start of one bridge to the start of the next.

Bridge Settings

Height (Only defined for 3D bridges.)

Angle (Only defined for 3D bridges.)

Start and end angle for 3D bridges. (0 deg = No slope, 90 deg = Vertical)

Length

Length of the bridge.

Type

2D or 3D.

2D bridges lift the cutter or turn it off and then restart at the end of the bridge. 2D bridges are generally appropriate for plasma and similar tools. 3D bridges raise the cutter to a specific height and then lower it again, without pausing the cutting process. Both the ascent and descent raise at the specified slope. 3D bridges are generally appropriate for routers.

Height Control

AVC Volts

Plasma power setting.

Note: Travelling Speed and Travelling Height global settings can be configured through "Options | Preferences | General Settings".

Choose Offset Toolpathing Parameters

Available templates

- 1: Alignment Laser
- 2: Flame Cutter
 - MILD STEEL
 - 50mm
 - Default
- 3: MS 50amp O202
- 7: MS 260amp O2Air
- Unassigned

Selected toolpathing parameters

Tool changer pos: 2

Tool name: Flame Cutter

Material: MILD STEEL

Thickness: 50mm

Template: Default

1. Cutting Settings

Material Name	MILD STEEL
Material Thickness	50
Tool Name	Flame Cutter
Type	Plasma Torch
Cut Width	4.0000
Cut Height	1.5000
Pierce Delay	2.00
XY FeedRate	2.400.0000
Pierce Height	75.0000
Overburn / Underburn	0.0000
Has Sharp Corners?	Yes
Reverse Direction	False
TrueHole Spiral %	0

2. Open Path Toolpathing

Apply toolpath to open contours?	No
Offset Direction	Centre
Collinear Leadin	No

3. Lead-in / Lead-out

Lead-in Type	Straight
Lead-in Length	5.0000
Lead-in Angle	90.00
Lead-out Type	None

4. Pre-Piercing

Perform Pre-Piercing?	Yes
Pre-Piercing Tool	MS 260amp O2Air
Type	Plasma Torch
Cut Height	1.5000
Pierce Delay	20.00
Pierce Height	75.0000
AVC Volts	170
Pre-piercing Offset	0.0000

1. Cutting Settings

Add Template

Save

Save As

Delete Template

OK

Cancel

Figure: Example of toolpathing parameters for thick material cutting. Plasma cutter is used for pre-piercing and then Flame cutter for the actual cutting.

Router Toolpathing Settings

1. Cutting Settings

Material Name

The name of the material.



Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Router

Cut Width

The width of the cut.

Depth

The depth of the cut.

(ActualDepthPerPass) This value is calculated automatically.

(DepthOfFirstPass) This value is calculated automatically.

(MaximumDepthPerPass) This value is calculated automatically.

Relative Depth

Input the depth relative to the "RelativeLocation" value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom

Below Surface

Above Bottom

XY Feed Rate

The XY feed rate.

Z Feed Rate

The Z feed rate.

Dwell

The time taken before the tool moves, after piercing the material.

Spindle Speed

The speed of the spindle.

Overcut / Undercut

This setting can be used to configure how cutting is finished for individual shapes. Define overcut by entering a positive value in and undercut by entering a negative value. The unit is mm. Default value is zero.

Ramp Leadin?

Select Yes or No. This setting is for tools that cannot plunge vertically.

Conventional / Climb

Climb milling and conventional milling are terms used to describe the direction in which the shape is cut. If the external edge of the shape is being cut out in an anticlockwise direction, this is called conventional milling. If the external edge of the shape is being cut out in a clockwise direction, this is called climb milling.

Has Sharp Corners?

Specify whether sharp corners are used in toolpathing. Default setting is Yes.

2. Open Path Toolpathing

Apply toolpath to open contours?

Select Yes or No.

Offset Direction

Centre: The toolpath will follow the path of the open-contour.

Left: The toolpath will provide the cutting path to the left of the open-contour.

Right: The toolpath will provide the cutting path to the right of the open-contour.

Colinear Leadin

Performs a reverse-direction leadin along the eventual toolpath. For short but deep toolpaths, it may need to zig-zag downwards. Not recommended for plasmas. Default value is false.

3. Lead-In / Lead-out

Lead-in Type

Select Straight, Arc, FollowTheSide or None.

Lead-in Length

The length of the lead-in.

Lead-in Angle

The angle of the lead-in.

Lead-out Type

Default value is None.

4. Pre-Piercing

Perform Pre-Piercing?

Specify whether pre-piercing is performed.

5. Multiple-Pass Settings

Number of Passes.

Define the number of passes to reach the final depth. This setting can override "Maximum depth per pass" setting. For example, the user can add the depth and the "Number of passes" will automatically adjust based on "Maximum depth per pass". But the user can then adjust the number of passes themselves ignoring the "Maximum depth per pass".

Ramping Settings

Separate Passes

Finishes one pass at a time returning to start point between passes.

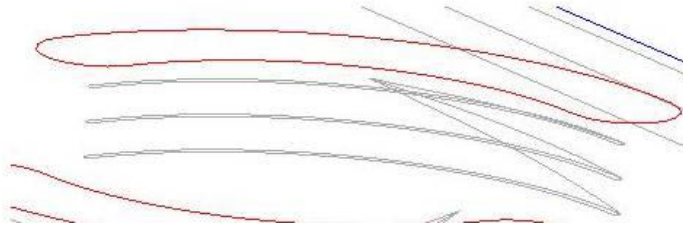


Figure: Example of using “Separate Passes” option when offset toolpathing a kidney shaped hole.

Ramp Down From Pass To Pass

After cutting a pass it will then proceed to ramp down to the next deeper pass by using the “Ramping Angle” value.

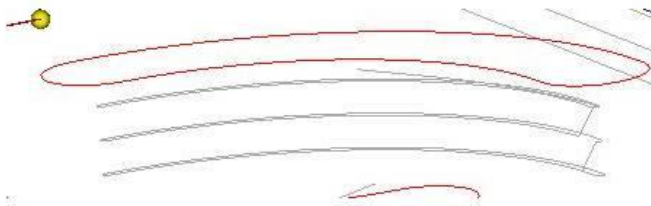


Figure: Example of using “Ramp Down From Pass To Pass” option when offset toolpathing a kidney shaped hole.

Continuous Ramping

Cuts at a constant angle so that exactly the same thickness is continually being removed from the material. It then makes one final cycle at the bottom to remove the remaining material. (This is same as “Continuous Spiral” option in previous ToolShop versions.)

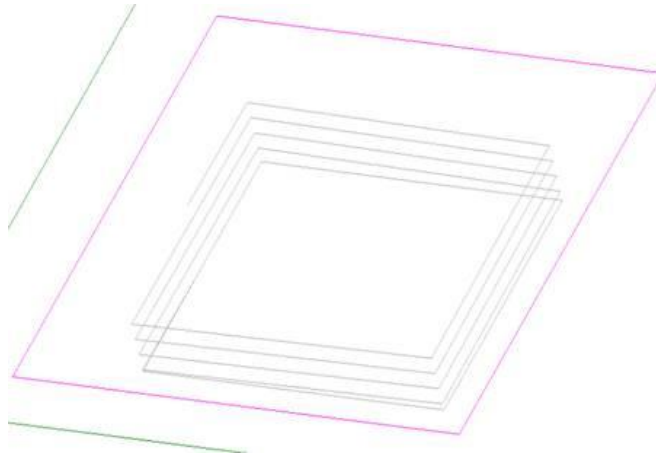


Figure: Example of using “Continuous Ramping” option when offset toolpathing a square hole.

Ramping Angle

Define the angle at which the cutter ramps down from one pass to the next pass.
Default value is 45 degrees.

Maximum Depth Per Pass

Define the maximum depth per pass. This setting automatically adjusts the number of passes if the depth is changed.

Use a Cleanup Pass?

Select Yes or No.

Cleanup Offset

Primary cut is offset this amount from the shape. The Cleanup pass removes this material and polishes the edge of the part.

Cleanup Pass Thickness

Primary cut path leaves this amount of material at the bottom of the cut.
Cleanup pass removes this material.

Remove Bridges?

Yes or No.

Starting Depth

Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitted but have the opposite effect. Default value is 0.

Relative Depth

Input here the starting depth.

Relative Location

Location from where the "Starting Depth" is calculated from.

Below Bottom
Below Surface
Above Bottom

6. Miscellaneous Settings**Place Bridges at Regular Intervals?**

Specify whether bridges will be automatically added.

Bridge Repeat Distance

The distance from the start of one bridge to the start of the next.

Bridge Settings

Height (Only defined for 3D bridges.)

Angle (Only defined for 3D bridges.)

Start and end angle for 3D bridges. (0 deg = No slope, 90 deg = Vertical)

Length

Length of the bridge.

Type

2D or 3D.

2D bridges lift the cutter or turn it off and then restart at the end of the bridge. 2D bridges are generally appropriate for plasma and similar tools. 3D bridges raise the cutter to a specific height and then lower it again, without pausing the cutting process. Both the ascent and descent raise at the specified slope. 3D bridges are generally appropriate for routers.

7. Inlay**Toolpath for Inlay?**

Select Yes or No. This option modifies the corners of the part so that inlay pieces will fit into cut out. This option should be used for male and female parts.

Inlay Radius

This specifies the amount of rounding for inlay corners. This should be equal or larger than the cutter radius. The same setting must be used for male and female parts.

Gap

Overlap between inner and outer toolpaths.

Note: Travelling Speed and Travelling Height global settings can be configured through "Options | Preferences | General Settings".

Choose Offset Toolpathing Parameters

Available templates

- 1: 6mm Router
 - MILD STEEL
 - 25mm
 - Default
 - Unassigned

Selected toolpathing parameters

Tool changer pos: 1

Tool name: 6mm Router

Material: MILD STEEL

Thickness: 25mm

Template: Default

Add Template

Save

Save As

Delete Template

1. Cutting Settings

Material Name: MILD STEEL

Material Thickness: 25

Tool Name: 6mm Router

Type: Router

Cut Width: 6.0000

Depth: 5.0000mm (Below Surface)

XY FeedRate: 4,500.0000

Z FeedRate: 2,500.0000

Dwell: 0.00

Spindle Speed: 18,000.00

Overcut / Undercut: 0.0000

Ramp Leadin: Yes

Conventional / Climb: Conventional

Has Sharp Corners?: Yes

2. Open Path Toolpathing

Apply toolpath to open contours? No

Offset Direction: Centre

Collinear Leadin: No

3. Lead-in / Lead-out

Lead-in Type: Straight

Lead-in Length: 5.0000

Lead-in Angle: 90.00

Lead-out Type: None

4. Pre-Piercing

Perform Pre-Piercing? No

5. Multiple-Pass Settings

Number of Passes: 3

Ramping Settings: Separate Passes

Ramping Angle: 45.00

Maximum depth per pass: 2.000

Use a Cleanup Pass? No

Starting Depth: 0.0000mm (Below Surface)

6. Miscellaneous Settings

Place Bridges at Regular Interval: No

7. Inlay

Toolpath for Inlay? No

Material Name

OK

Cancel

Figure: Example of toolpathing parameters for a router tool.



Centre

The settings for centre-line cutting are same as for offset cutting. However with centre-line cutting the tool follows the centre of the toolpath. This is same as defining tool offset as zero.



Fill

This function opens the “Choose Fill Toolpathing Parameters” window which allows you to add fills to selected shapes.

Fill Settings

1. Cutting Settings

Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Router

Cut Width

The width of the cut

Depth

The depth of the cut.

(ActualDepthPerPass) This value is calculated automatically.

(DepthOfFirstPass) This value is calculated automatically.

(MaximumDepthPerPass) This value is calculated automatically.

Relative Depth

Input the depth relative to the “RelativeLocation” value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom

Below Surface

Above Bottom

XY Feed Rate

Feed rate at which the machine will travel while it is cutting through the material. This does not affect the rapid feed rate of the tool when it travels above the surface of the material.

Z Feed Rate

Feed rate at which the machine will plunge into the material in a vertical direction. This does not affect the XY feed rate or the rapid feed rate of the tool when it travels above the surface of the material.

Dwell

The dwell value denotes how long the machine will pause at the end of the first plunge of each shape. Its unit of measurement is seconds.

Spindle Speed

Spindle RPM

Conventional / Climb

Conventional and climb milling are terms used to describe the direction in which the shape is cut. If the external edge of the shape is being cut out in an anticlockwise direction, this is called conventional milling. If the external edge of the shape is being cut out in a clockwise direction, this is called climb milling.

Has Sharp Corners?

Specify whether sharp corners are used in toolpathing. Default setting is Yes.

2. Fill Settings

Overlap Percentage

Default value is 10.

Minimize Lifting the Cutter Head

Self-explanatory

Fill Pattern

Hatch fill or Island fill.

Fill angle (degrees)

This is the angle for linear fill pattern. Default value is 0.

Has a Ramped Entry?

Select Yes or No. Determines whether the cutting head ramps into the material or enters vertically.

Ramping Angle (degrees)

The downwards angle at which the cutting head should start any new engraving incision. Default value is 45.

5. Multiple-Pass Settings

Number of Passes

Default value is 1.

Maximum Depth Per Pass

Define maximum depth per pass. This setting automatically adjusts the number of passes if it is changed.

Use a Cleanup Pass?
Select Yes or No.

Cleanup Offset
Primary cut is offset this amount from the shape. The Cleanup pass removes this material and polishes the edge of the part.

Cleanup Pass Thickness
Primary cut path leaves this amount of material at the bottom of the cut. Cleanup pass removes this material.

Starting Depth
Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitted but have the opposite effect. Default value is 0.

Relative Depth
Input here the starting depth.

Relative Location
Location from where the "Starting Depth" is calculated from.

Below Bottom
Below Surface
Above Bottom

7. Inlay

Toolpath for Inlay?
Select Yes or No. This option modifies the corners of the part so that inlay pieces will fit into cut out. This option should be used for male and female parts.

Inlay Chamfer Radius
This specifies the amount of rounding for inlay corners. This should be equal or larger than the cutter radius. The same setting must be used for male and female parts.

Overlap thickness
Overlap between inner and outer toolpaths

Note: *Travelling Speed and Travelling Height* global settings can be configured through "Options | Preferences | General Settings."

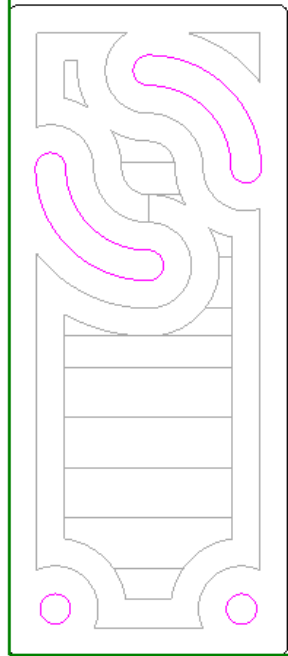


Figure: A picture of a shape that was linear filled by using 12mm wide routing tool and 180 degree fill angle

Note: If user has selected Inlay toolpathing and if the “**Inlay Chamfer Radius**” is smaller than the radius of the cutter then ToolShop shows a warning.

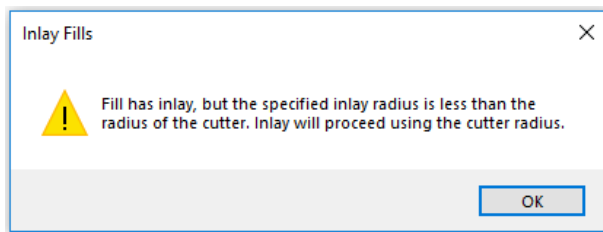


Figure: If user is toolpathing for inlay and the “**Inlay Chamfer Radius**” is smaller than the radius of the cutter then the radius of the cutter is used instead.

Choose Fill Toolpathing Parameters

Available templates

- 1: 6mm Router
 - MILD STEEL
 - 19.05mm
 - Default
 - Unassigned

Selected toolpathing parameters

Tool changer pos: 1

Tool name: 6mm Router

Material: MILD STEEL

Thickness: 19.05mm

Template: Default

Add Template

Save

Save As

Delete Template

1. Cutting Settings

Material Name: MILD STEEL

Material Thickness: 19.05

Tool Name: 6mm Router

Type: Router

Cut Width: 6.0000

Depth: 5.0000mm (Below Surface)

XY FeedRate: 4.500.0000

Z FeedRate: 2.500.0000

Dwell: 0.00

Spindle Speed: 18.000.00

Conventional / Climb: Conventional

Has Sharp Corners?: Yes

2. Fill Settings

Overlap Percentage: 10

Minimize Lifting the Cutter Head: No

Fill Pattern: Hatch Fill

Fill Angle (degrees): 45.00

Has a Ramped Entry?: Yes

Ramping Angle (degrees): 45.00

5. Multiple-Pass Settings

Number of Passes: 2

Maximum depth per pass: 3.000

Use a Cleanup Pass?: No

Starting Depth: 0 (Below Surface)

RelativeDepth: 0.0000

RelativeLocation: Below Surface

7. Inlay

Toolpath for Inlay?: No

Starting Depth

Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitte...

OK

Cancel

Figure: Typical settings for hatch filling.



Thread Milling

This function opens the “Choose Thread Milling Parameters” window which allows you to add threading to holes.

Thread Milling Settings

1. Cutting Settings

Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Thread Milling

Cutter Diameter

The diameter of the cutter

Thread Pitch

The pitch of a thread in a tapped hole.

Thread Cut-in Depth

The depth at which thread cutting starts.

Taper Thread?

Select Yes or No. Default value is No. Tapered threads are sealing threads.

Feed Rate

Feed rate at which the machine will plunge into the material in a vertical direction. This does not affect the XY feed rate or the rapid feed rate of the tool when it travels above the surface of the material.

RPM

Spindle RPM.

Has a right-hand thread?

Select Yes or No. Default value is Yes. By common convention, right-handedness is the default handedness for screw threads.

Cut outside the circle?

Select Yes or No. Default value is No. Select whether to cut outside the circle.

Conventional / Climb

Conventional and climb milling are terms used to describe the direction in which the shape is cut. If the external edge of the shape is being cut out in an anticlockwise direction, this is called conventional milling. If the external edge of the shape is being cut out in a clockwise direction, this is called climb milling. Default value is Conventional.

Cut Downwards?

Select Yes or No. Default value is Yes.

2. Multiple-Pass Settings

Starting Depth

Starting depth of the thread.

Finishing Depth

Finishing depth of the thread.

Number of Passes

Default value is 1.

Use a Cleanup Pass?

Select Yes or No. Default value is No.

Cleanup Pass Thickness

Primary cut path leaves this amount of material at the bottom of the cut. Cleanup pass removes this material.

Maximum Cut-in Depth Per Pass

Define maximum cut-in depth per pass. This setting automatically adjusts the number of passes if it is changed.

3. Lead-In / Lead-out**Lead-in Type**

Select Straight, Arc, FollowTheSide or None.

Lead-in Length

The length of the lead-in.

Lead-in Angle

The angle of the lead-in.

Lead-out Type

Default value is None.

4. Pre-Fill**Has Pre-Fill?**

Select Yes or No.

Pre-Fill Template

Select the pre-fill template.

Note: *Travelling Speed* and *Travelling Height* global settings can be configured through "Options | Preferences | General Settings".

Choose Thread Milling Parameters

Available templates

m

4: ThreadMill

MILD STEEL

20mm

Default

Selected toolpathing parameters

Tool changer pos: 4

Tool name: ThreadMill

Material: MILD STEEL

Thickness: 20mm

Template: Default

Add Template

Save

Save As

Delete Template

1. Cutting Settings

Material Name

MILD STEEL

Material Thickness

20

Tool Name

ThreadMill

Type

Thread Milling

Cutter Diameter

5.0000

Thread Pitch

5.000

Thread Cut-in Depth

1.000

Taper Thread?

No

Feed Rate

2,456.0000

RPM

567

Has a right-hand thread?

Yes

Cut outside the circle?

No

Conventional / Climb

Conventional

Cut Downwards?

Yes

2. Multiple-Pass Settings

Starting Depth

-1.000

Finishing Depth

10.000

Number of Passes

1

Use a Cleanup Pass?

Yes

Cleanup Pass Thickness

0.1000

Maximum Cut-in depth Per Pass

0

3. Lead-in / Lead-out

Lead-in Type

Straight

Lead-in Length

0.1000

Lead-in Angle

90.00

Lead-out Type

None

4. Pre-Fill

Has Pre-Fill?

Yes

Prefill Template

<default>

Material Name

OK

Cancel

Figure: Typical settings for Thread Milling.

Marking

This function opens the “Choose Marking Parameters” dialog box which allows user to add a marking toolpath to the selected shapes.

Marking Settings

1. Cutting Settings
- Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Router

Cut Width

The width of the cut

Depth

The depth of the cut.

XY Feed Rate

Feed rate at which the machine will travel while it is cutting through the material. This does not affect the rapid feed rate of the tool when it travels above the surface of the material.

Z Feed Rate

Feed rate at which the machine will plunge into the material in a vertical direction. This does not affect the XY feed rate or the rapid feed rate of the tool when it travels above the surface of the material.

Dwell

The dwell value denotes how long the machine will pause at the end of the first plunge of each shape. Its unit of measurement is seconds.

Spindle Speed

Spindle RPM

Overcut / Undercut

This setting can be used to configure how cutting is finished for individual shapes. Define overcut by entering a positive value in and undercut by entering a negative value. The unit is mm. Default value is zero.

Ramp Leadin?

Select Yes or No. This setting is for tools that cannot plunge vertically.

Conventional / Climb

Climb milling and conventional milling are terms used to describe the direction in which the shape is cut. If the external edge of the shape is being cut out in an anticlockwise direction, this is called conventional milling. If the external edge of the shape is being cut out in a clockwise direction, this is called climb milling.

Has Sharp Corners?

Specify whether sharp corners are used in toolpathing. Default setting is Yes.

2. Open Path Toolpathing

Apply toolpath to open contours?

Select Yes or No.

Offset Direction

Centre: The toolpath will follow the path of the open-contour.

Left: The toolpath will provide the cutting path to the left of the open-contour.

Right: The toolpath will provide the cutting path to the right of the open-contour.

Colinear Leadin

Performs a reverse-direction leadin along the eventual toolpath. For short but deep toolpaths, it may need to zig-zag downwards. Not recommended for plasmas. Default value is false.

3. Lead-In / Lead-Out

Lead-in Type

Select Straight, Arc, FollowTheSide or None.

Lead-in Length

The length of the lead-in.

Lead-in Angle

The angle of the lead-in.

Lead-out Type

Default value is None.

4. Pre-Piercing

Perform Pre-Piercing?

Specify whether pre-piercing is performed.

5. Multiple-Pass Settings

Number of Passes.

Define the number of passes to reach the final depth. This setting can override "Maximum depth per pass" setting. For example, the user can add the depth and the "Number of passes" will automatically adjust based on "Maximum depth per pass". But the user can then adjust the number of passes themselves ignoring the "Maximum depth per pass".

Ramping Settings

Separate Passes

Finishes one pass at a time returning to start point between passes.

Ramp Down From Pass To Pass

After cutting a pass it will then proceed to ramp down to the next deeper pass by using the "Ramping Angle" value.

Continuous Ramping

Cuts at a constant angle so that exactly the same thickness is continually being removed from the material. It then makes one final cycle at the bottom to remove the remaining material. (This is same as "Continuous Spiral" option in previous ToolShop versions.)

Ramping Angle

Define the angle at which the cutter ramps down from one pass to the next pass. Default value is 45 degrees.

Maximum Depth Per Pass

Define the maximum depth per pass. This setting automatically adjusts the number of passes if the depth is changed.

Use a Cleanup Pass?

Select Yes or No.

Cleanup Offset

Primary cut is offset this amount from the shape. The Cleanup pass removes this material and polishes the edge of the part.

Cleanup Pass Thickness

Primary cut path leaves this amount of material at the bottom of the cut. Cleanup pass removes this material.

Remove Bridges?

Yes or No.

Starting Depth

Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitted but have the opposite effect. Default value is 0.

Relative Depth

Input here the starting depth.

Relative Location

Location from where the "Starting Depth" is calculated from.

Below Bottom

Below Surface

Above Bottom

6. Miscellaneous Settings**Place Bridges at Regular Intervals?**

Specify whether bridges will be automatically added.

7. Inlay**Toolpath for Inlay?**

Select Yes or No. This option modifies the corners of the part so that inlay pieces will fit into cut out. This option should be used for male and female parts.

Inlay Radius

This specifies the amount of rounding for inlay corners. This should be equal or larger than the cutter radius. The same setting must be used for male and female parts.

Gap

Overlap between inner and outer toolpaths.

Commented [JR10]: added

**Diamond (3D Engraving)**

This function opens the “Choose Diamond Cut Toolpathing Parameters” dialog box which allows user to add a diamond cut toolpath to the selected shapes. This feature uses V-Cutters to create sharp corners and a 3D look.

Diamond Cut Settings**1. Cutting Settings****Material Name**

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Router V Cutter

Cut Width

The width of the cut

Depth

The depth of the cut.

(ActualDepthPerPass) This value is calculated automatically.

(DepthOfFirstPass) This value is calculated automatically.

(MaximumDepthPerPass) This value is calculated automatically.

Relative Depth

Input the depth relative to the “RelativeLocation” value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom

Below Surface

Above Bottom

XY Feed Rate

Feed rate at which the machine will travel while it is cutting through the material. This does not affect the rapid feed rate of the tool when it travels above the surface of the material.

Z Feed Rate

Feed rate at which the machine will plunge into the material in a vertical direction. This does not affect the XY feed rate or the rapid feed rate of the tool when it travels above the surface of the material.

Dwell

The dwell value denotes how long the machine will pause at the end of the first plunge of each shape. Its unit of measurement is seconds.

Spindle Speed

Spindle RPM

Conventional / Climb

Conventional and climb milling are terms used to describe the direction in which the shape is cut. If the external edge of the shape is being cut out in an anticlockwise direction, this is called conventional milling. If the external edge of the shape is being cut out in a clockwise direction, this is called climb milling. Default value is Conventional.

2. Fill Settings

Overlap Percentage

Default value is 10.

Minimize Lifting the Cutter Head

Self-explanatory

Fill Pattern

Hatch fill or Island fill.

Fill angle (degrees)

This is the angle for linear fill pattern. Default value is 0.

Cut angle (degrees)

The cut angle of V cutter, eg. 90deg cutter profile has 45deg cut angle. Default value is 45.

Corner type

Sharp
Clipped
Round

5. Multiple-Pass Settings

Number of Passes

Default value is 1.

Maximum Depth Per Pass

Define maximum depth per pass. This setting automatically adjusts the number of passes if it is changed.



Starting Depth

Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitted, but have the opposite effect.

Relative Depth

Input the depth relative to the "RelativeLocation" value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom
Below Surface
Above Bottom

Choose Diamond Cut Toolpathing Parameters

Available templates

- 1: 90Deg V Cutter
 - MILD STEEL
 - 20mm
 - Default
 - Unassigned

Selected toolpathing parameters

Tool changer pos: 1

Tool name: 90Deg V Cutter

Material: MILD STEEL

Thickness: 20mm

Template: Default

Add Template

Save

Save As

Delete Template

1. Cutting Settings

Material Name	MILD STEEL
Material Thickness	20
Tool Name	90Deg V Cutter
Type	Router V Cutter
Cut Width	16.0000
Depth	5.0000mm (Below Surface)
ActualDepthPerPass	2.5000
DepthOfFirstPass	2.5000
MaximumDepthPerPass	3.0000
RelativeDepth	5.0000
RelativeLocation	Below Surface
XY FeedRate	2,000.0000
Z FeedRate	2,000.0000
Dwell	0.00
Spindle Speed	18,000.00
Conventional / Climb	Conventional

2. Fill Settings

Overlap Percentage	10
Minimize Lifting the Cutter Head	No
Fill Pattern	Hatch Fill
Fill Angle (degrees)	0.00
Cut Angle (degrees)	45.00
Corner Type	Sharp

5. Multiple-Pass Settings

Number of Passes	2
Maximum depth per pass	3.000
Starting Depth	0 (Below Surface)
RelativeDepth	0.0000
RelativeLocation	Below Surface

Material Name

OK

Cancel

Figure: Typical settings for diamond cutting.



Drill / Pierce

This function opens the "Choose Drill/Piercing Toolpathing Parameters" dialog box which allows user to activate the drill point placement mode or to toolpath selected closed shapes for drill or piercing. If you are toolpathing selected shapes for drill they must have been previously processed using the "Cleanup" and "Combine" functions for this function to work correctly.

Choose the drill settings you wish to use to toolpath selected shapes or to use in the drill point placement mode.

Drill Toolpathing Settings

1. Cutting Settings

Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Drill

Drill Diameter

The diameter of drill.

Depth

The distance to drill.

(ActualDepthPerPass) This value is calculated automatically.

(DepthOfFirstPass) This value is calculated automatically.

(MaximumDepthPerPass) This value is calculated automatically.

Relative Depth

Input the depth relative to the "RelativeLocation" value. This will allow you to use a single toolpathing template for various material thicknesses.

RelativeLocation

Location from where the depth is calculated from.

Below Bottom

Below Surface

Above Bottom

Z Feed Rate

Plunge speed.

Dwell

The time taken before the tool moves, after piercing the material.

Drill RPM

Drill revolutions per minute.

5. Multiple-Pass Settings

Starting Depth

Starting depth of the first pass or peck. The distance below the surface that the primary toolpath starts. Negative values are permitted, but have the opposite effect.

Number of Pecks

Specify the number of drill pecks.

Peck Retract Distance

Distance the drill retracts after each peck. This is an offset value from tool's current depth.

Note: If you wish the 'Peck Retract Distance' to be an absolute value from the material surface then set the option '**Retract above surface**' to True in "Options | Preferences | Toolpathing Settings".

Note: Travelling Speed and Travelling Height global settings can be configured through "Options | Preferences | General Settings".

Choose Drill / Piercing Toolpathing Parameters

Available templates

- 2: 6.0mm Drill
 - MILD STEEL
 - 15mm
 - MS 6mm Drill

Selected toolpathing parameters

Tool changer pos: 2

Tool name: 6.0mm Drill

Material: MILD STEEL

Thickness: 15mm

Template: MS 6mm Drill

1. Cutting Settings

Material Name: MILD STEEL

Material Thickness: 15

Tool Name: 6.0mm Drill

Type: Drill

Drill Diameter: 6.000

Depth: 5.000mm (Below Surface)

ActualDepthPerPass: 5.000

DepthOfFirstPass: 0.000

MaximumDepthPerPass: 0.000

RelativeDepth: 5.000

RelativeLocation: Below Surface

Z FeedRate: 1,200.000

Dwell: 0.00

Drill RPM: 800

5. Multiple-Pass Settings

Starting Depth: -5.000mm (Below Surface)

RelativeDepth: -5.000

RelativeLocation: Below Surface

Number of Pecks: 2

Peck Retract Distance: 0.000

Material Name

Add Template

Save

Save As

Delete Template

Place Drill Points

OK

Cancel

Figure: Toolpathing Parameters dialog box for drill toolpath.

Drill Point Placement Mode

After you have selected the appropriate drill settings click on the “Place Drill Points” button on the “Choose Drill/Piercing Toolpathing Parameters” dialog box to enter drill point placement mode. The mouse cursor changes to cross-hair cursor indicating a valid placement. You can now start placing the drill points by left mouse clicking on the desired locations. If you move the mouse cursor near the edge of an existing drill point then the mouse cursor changes to select cursor which allows you to select the drill point.

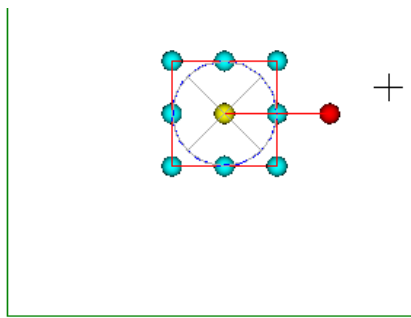


Figure: Placing drill points by left mouse clicking.

Change the drill settings while placing drill points via drill point placement popup menu. To access the popup menu right click anywhere on the screen. After modifying the settings you may accept them by clicking “OK” button or you may discard them by clicking “Cancel” button. Clicking either button will return you back to the drill point placement mode.

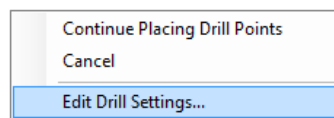


Figure: During the drill point placement you can change the drill settings via a popup menu.

Continue Placing Drill Points

Returns to the Drill Point Placement mode where you can add new drill points.

Cancel

Cancels the Drill Point Placement mode and switches ToolShop back to the main Mouse Selection mode.

Edit Drill Settings

Goes back to the “Edit Drill Settings” dialog where you can change the drill settings.

Note: While you are in the Drill Point Placement mode you can remove any selected drill point by pressing the Del key on the keyboard.



Print Head

This function specifically supports D.O.D. style inkjet text printers with or without a rotating axis. Most ART Routing and Plasma cutting machines may be fitted with this device for marking part numbers and fold lines etc. Other brands of machines may be supported if this type of accessory is available for that model.

To use this function first select text objects on the screen. Click the Print Head toolpath button to display the toolpath dialog box.

Note: As most text printers have limited font sizes, these text objects should already be drawn in the CAD program at the correct height to match the printer capabilities. This is important so that ToolShop can display an accurate layout on the screen i.e. if the printer prints 14mm high letters, then the drawing should have 14mm high text.

Print Head Toolpathing Settings

1. Cutting Settings

Material Name

The name of the material.

Material Thickness

The thickness of the material.

Tool Name

The name of the current tool

Type

Inkjet Printer

Text Height

Text height should be same as the printer font.

XY Feed Rate

XY feed rate should match the maximum printing speed. (Usually around 4000)

Text Height/Width Ratio

The ratio should be the printer character width / character height .
(Usually around 0.50)

Add Extra Spaces

This is a small buffer to ensure the baseline is longer than the printed text.
(Usually 2 extra spaces)

Note: These values normally do not change from job to job. The defaults should be setup correctly in the "Options | Tool Settings" dialog box.

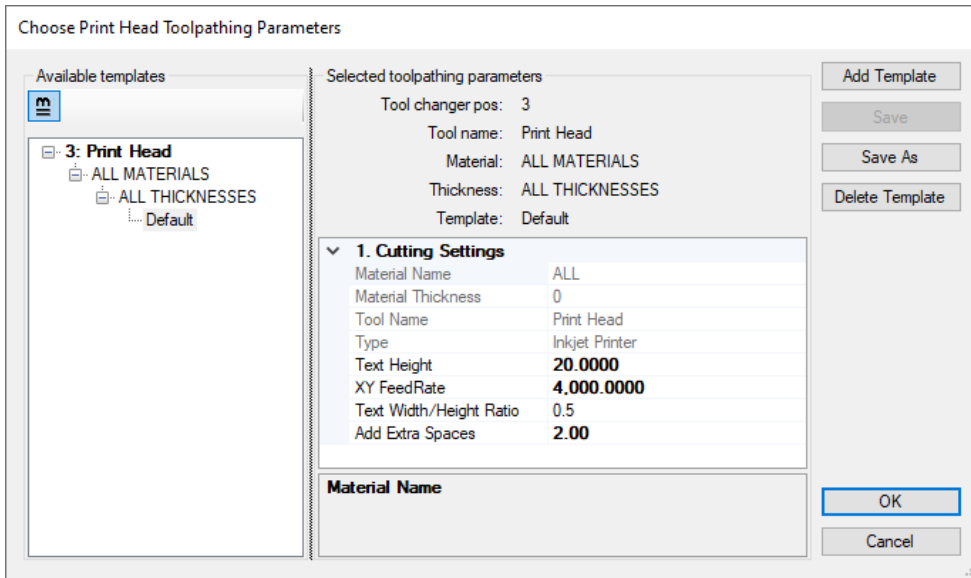


Figure: Typical settings for print head toolpathing.



Edit

This function shows a dialog box similar to “Choose Offset Toolpathing Parameters” dialog box and allows you to edit toolpathing parameters for selected shapes. After accepting the changes the objects are re-toolpathed.



Delete

This function is provided to delete previously generated toolpaths from selected objects. Use this option to alter the toolpath settings on an object or exclude a particular object from the CNC file output to the machine.



Multi Process

This function opens the “Choose Multi Process Toolpathing Settings” window which allows you to add multiple toolpaths to the selected shapes in a single call.

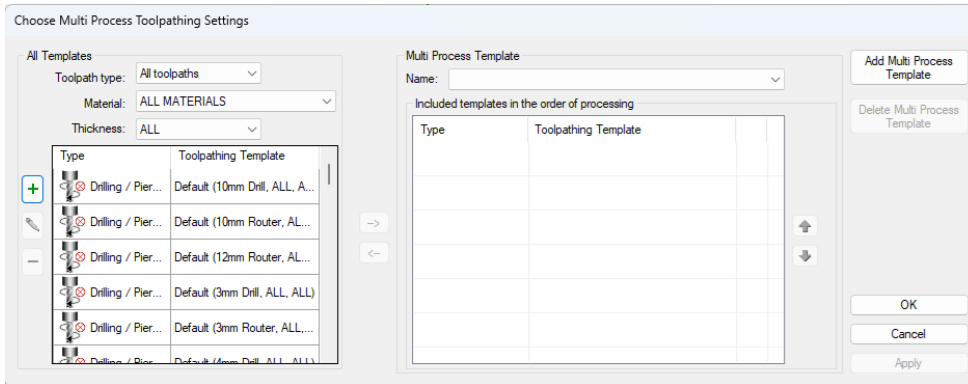


Figure: Multi process toolpathing window.

The multi process toolpathing window is divided into two sections:

- The “All Templates” section lists all currently available templates. You may narrow down the list by choosing from the available “Toolpath type”, “Material” and “Thickness” options.
- The “Multi Process Template” section has a drop-down list of all defined multi-process templates. Select an individual multi-process template from the list to see the included toolpathing templates.

After selecting the multi-process template, click “OK” button add the toolpaths to your selected shapes. The “Apply” button saves any changes into the active multi-process template. Use “Cancel” button to exit the function.



All Templates | Add new template

Click this button to add a new toolpathing template. Choose from the pop-up menu the type of toolpathing template you wish to add.

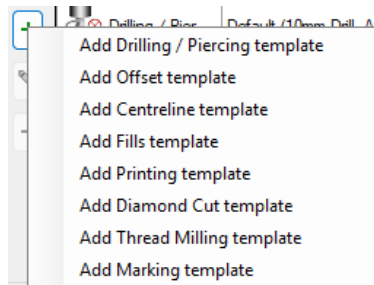


Figure: Choosing the template type.

This opens the “New Tool Template” window. Choose from the “Tool Name” list the tool you wish to use. The “Template Type” field shows the selected toolpathing template type.

Figure: Choose the tool, material and name for the new template.

If you want the template to apply to a specific material name, tick the “Apply to specific material name” check box and select the appropriate material from the drop-down list. If “Apply to specific material name” is unselected, then template material name “ALL” is used. If “Apply to specific thicknesses” is selected, enter the “Thickness” field the desired material thickness this template should apply to. If “Apply to specific thicknesses” is not selected, then the template material thickness is saved as “ALL”. Type into the “Template Name” field the desired name. You may also use the “Default” name. However, only one “Default” named template per toolpathing type, material and thickness combination is allowed.

Click “OK” button to add the new toolpathing template. Clicking “Cancel” button will exit from the “Add new template” function.



All Templates | Edit template

Click this button to edit the toolpathing parameters of the selected template.



All Templates | Delete template

Click this button to delete the selected template from the Tool Library.

Multi Process Template | Add Multi Process Template

Click “Add Multi Process Template” button to create a new multi-process template. This function will open the “Add Multi Process Template” window. Choose a name and whether the multi-process template should apply to a specific material. The material and thickness fields are set as “ALL” by default. Click “OK” to create the multi-process template.

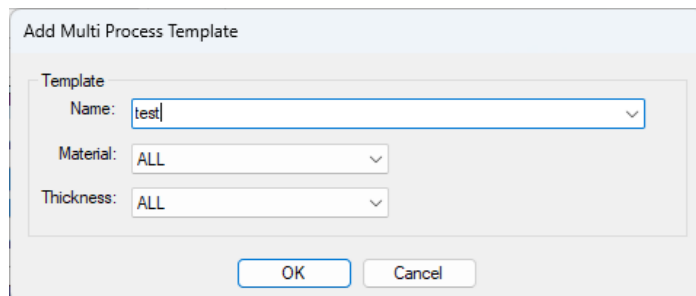


Figure: Adding a multi-process template.

After creating the multi-process template, you can add templates to the multi-process template by using the “Add” button. To remove a template, click the “Remove” button. If the multi-process template contains more than one template, you can modify the order of processing of those templates by using the “Up” and “Down” arrow buttons.

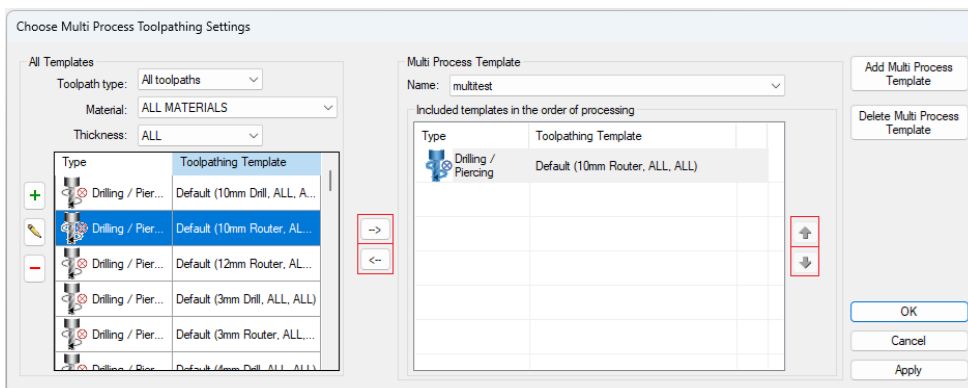


Figure: A multi-process template with one included “Drilling/Piercing” template.

Multi Process Template | Delete Multi Process Template

Click “Delete Multi Process Template” button to delete the currently selected multi-process template.



Link Cutting

Enabling this function allows the user to ‘link’ several shapes together to reduce the number of start points on plasma tools. This can dramatically increase the life of the consumables. While this is very desirable, it must be used carefully. Usually, several parts will be positioned around in a circle, or alternatively in a couple of rows. You must use the Lead-In and Lead-Out options with this function. Usually, a lead-in of 45 degrees and a lead-out of 45 degrees would be desirable. The Lead-Ins need to be positioned in the space between the parts to be linked together. After selecting the Link Cutting function you can click and hold the mouse on any lead-in. The program will automatically display any other start points that are compatible to be linked to the first shape. By dragging the mouse to the next desired start point and releasing

the mouse button, ToolShop will display a link between the two shapes. This may be repeated as many times as desired.

After two or more shapes are linked, they are locked together and may only be moved as a group. You may still edit the lead-in positions however.

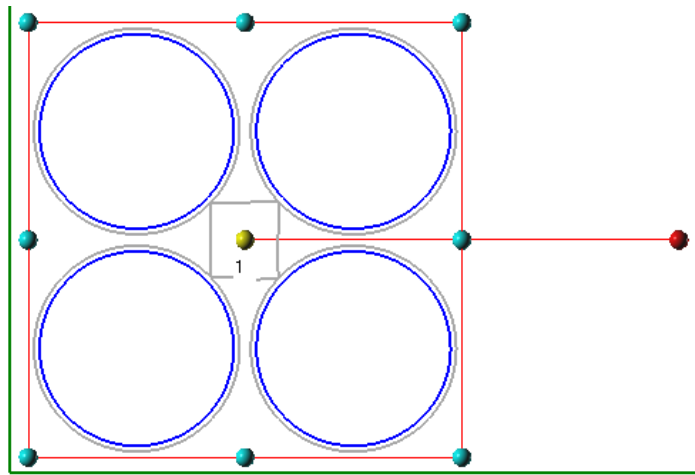


Figure: Using link cutting to cut 4 shapes. Both Lead-in and Lead-out are 45 degrees. Notice that the Lead-ins are positioned in the space between the parts to be linked together. By enabling the "View | Show Cut Order" from the menu you can also check the start point of the linked cutting.

Right mouse click on top of a link line. This will show you a popup menu that enables you to remove links between shapes.

Continue Linking Toolpaths

Goes back to the Link Cutting mode where you can add new links or remove existing links.

Cancel

Cancels the Link Cutting mode and switches ToolShop back to the main Mouse Selection mode.

Remove Link

Removes the link.

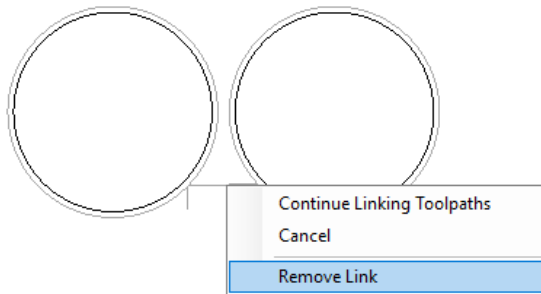


Figure: Removing a link.

Note: Delete Toolpath function will remove the links.



Bridges

This function enables bridges to be defined onto toolpathed shapes. First the function shows “Edit Bridge Settings” dialog allowing you to select the desired bridge settings. If the length of the bridge is greater than the perimeter of the shape, then the perimeter of the shape is used.

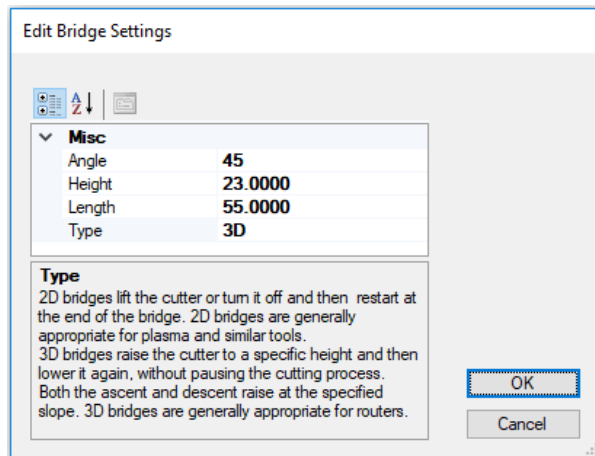


Figure: Defining a 3D bridge.

The “Edit Bridge Settings” options are as follows:

- **Angle.** (Only defined for 3D bridges)
 - Start and end angle for bridges. (0 deg = No slope, 90 deg = Vertical)
- **Height.** (Only defined for 3D bridges)
 - Height of the bridge.

- **Length.** Length of the bridge.
- **Type.** 2D or 3D.

2D bridges lift the cutter or turn it off and then restart at the end of the bridge. 2D bridges are generally appropriate for plasma and similar tools. 3D bridges raise the cutter to a specific height and then lower it again, without pausing the cutting process. Both the ascent and descent raise at the specified slope. 3D bridges are generally appropriate for routers.

After selecting the bridge settings click OK button to enter Bridge Placement mode. The mouse cursor changes into Bridge placement cursor when moved over any toolpath. Click left mouse button down on the toolpath to place the bridge.

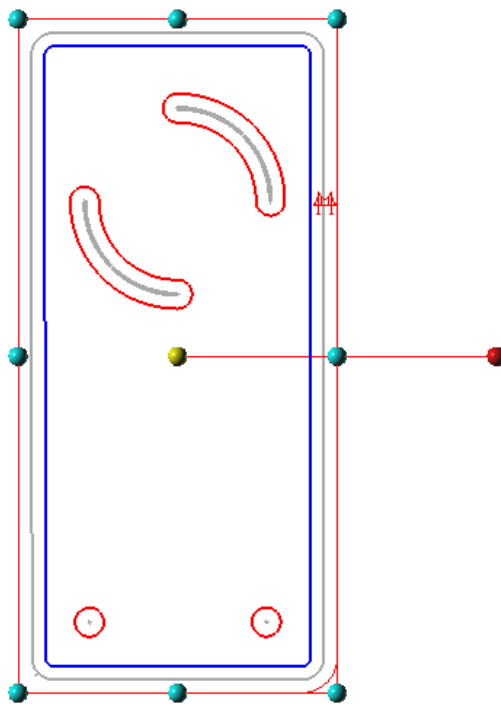


Figure: Place a bridge by left mouse clicking over the toolpath.

Move the mouse cursor over an existing bridge and the mouse cursor changes to Movement cursor. The end points of the bridge are also highlighted with small blue circles. Drag with the left mouse button pressed to move the bridge along the toolpath or you can drag it to another toolpath.

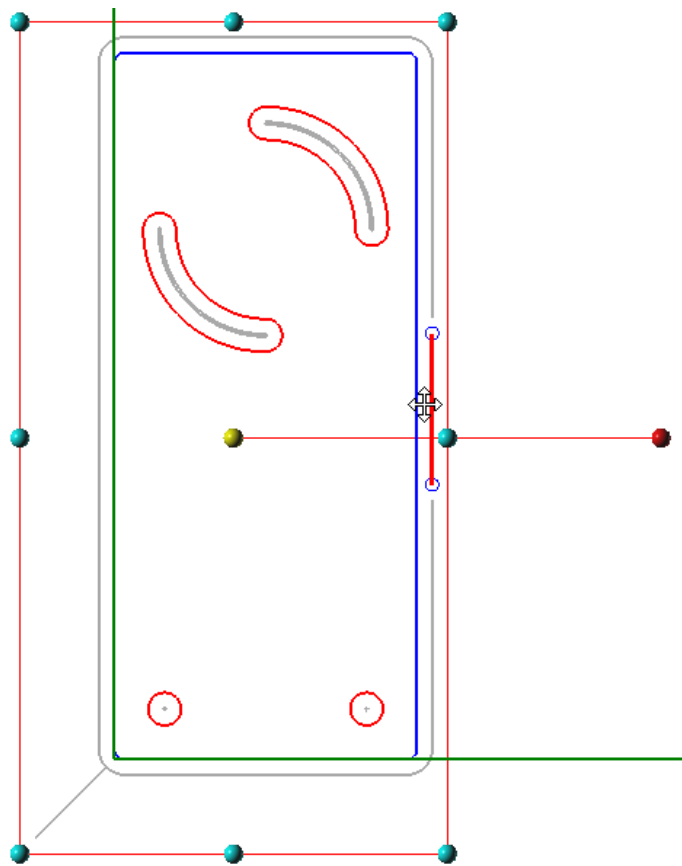


Figure: You can move the bridge by dragging with the left mouse button down.

To show a popup menu that enables you to remove or edit the bridge right mouse click on top of an existing bridge.

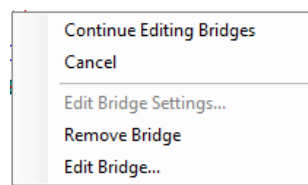


Figure: Bridge placement popup menu.

Continue Editing Bridges

Goes back to the bridge editing mode where you can add new bridges or edit and move existing bridges.

Cancel

Cancels the bridge editing and switches ToolShop back to the main Mouse Selection mode.

Edit Bridge Settings

To change the bridge settings during the bridge placement mode, right click anywhere on the screen to access the "Edit Bridge Settings" option.

Remove Bridge

Removes the bridge.

Edit Bridge

Allows user to adjust the properties of the bridge.

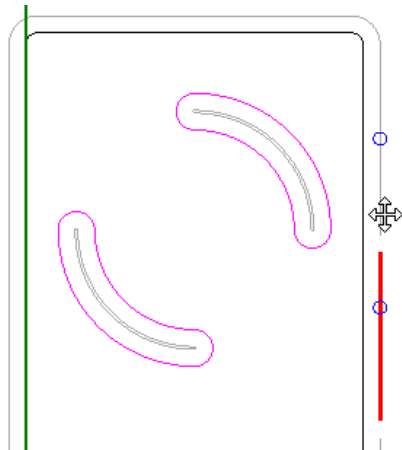


Figure: The bridge is shown as a red line on top of the toolpath line. When dragging the bridge two blue circles indicate the new position of the bridge.



Line of Bridges

Click this button to enable the Line of Bridges function. It enables you to draw a line of bridges across toolpathed shapes. First the function shows the "Line of Bridges | Edit Bridge Settings" dialog box allowing you to select the desired bridge settings.

After selecting the bridge settings click OK button to enter the Bridge Placement mode. The mouse cursor changes into a crosshair cursor. To draw a line of bridges, click left mouse button down at the selected start point and then at the end point. Each additional left mouse click adds a new line segment to create a single continuous line. To generate the bridges, right mouse click anywhere on the work sheet and then choose "Finish Line of Bridges" from the pop-up menu.

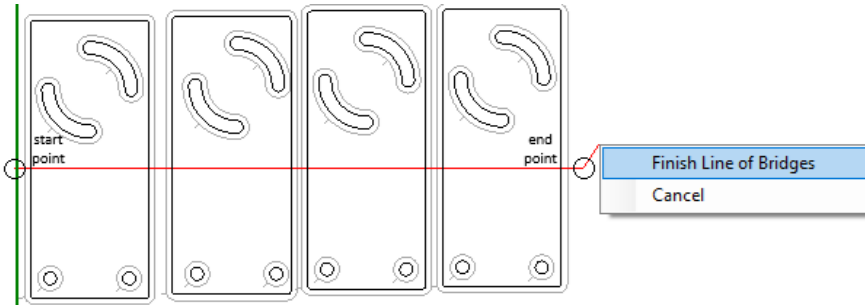


Figure: Draw line across the toolpathed shapes where you'd like the bridges to be placed.

The "Cancel" menu option cancels the operation and exits the Bridge Placement mode.

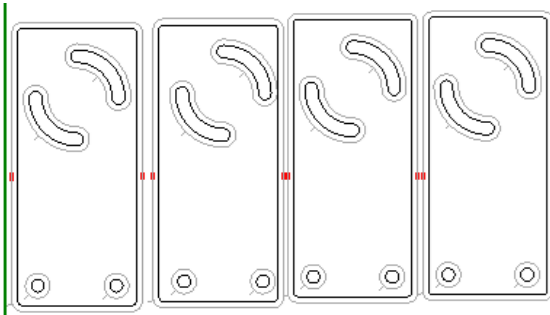
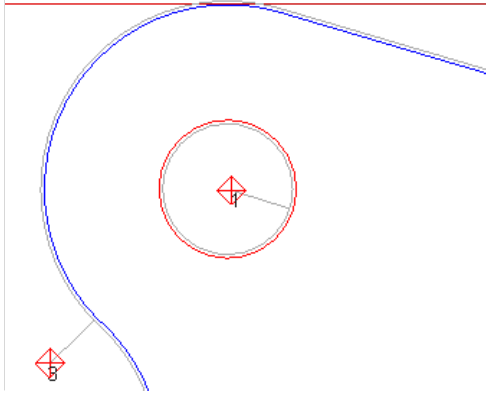


Figure: The bridges will be placed where the line crosses the toolpath.



Edit Start Points

Enabling this function will display a marker at the start position for each toolpath. Move each marker with the mouse by dragging it along the toolpath to a new location. Ensure that the position of the start point is not on top of another object. It is good practice to position the start points on a corner. Correct positioning of start points is critical for good machining. It is also useful to ensure that other shapes are not cut when piercing through the material. Perform this function after nesting.



**Figure: Start point markers indicating the lead-ins.
Move each marker with the mouse.**

Reset Start Points

The “Reset Start Points” buttons enable you to re-position the lead-ins and lead-outs. This is done after toolpathing and nesting. Select the shapes which start points you wish to reset and then click on the appropriate start point location button. If you wish ToolShop to reset the start points automatically after nesting, then select “Yes” for “Reset Lead-Ins and Lead-Outs To Direction” in “Choose Nesting Parameters” dialog box.

The available start point locations are:



Top

Re-position start points to top.



Bottom

Re-position start points to bottom.



Left

Re-position start points to left.



Right

Re-position start points to right.



Top Left

Re-position start points to top left.



Top Right

Re-position start points to top right.



Bottom Left

Re-position start points to bottom left.



Bottom Right

Re-position start points to bottom right.

Layout menu

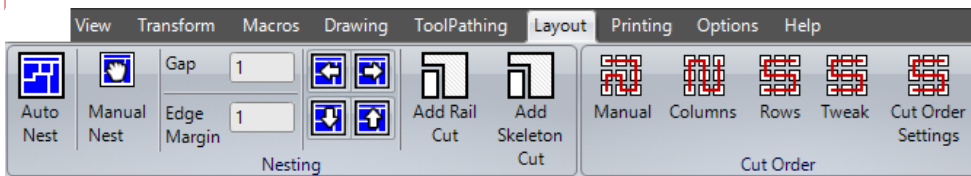


Figure: The Layout menu allows user to do nesting related operations.

Commented [JR11]: Pic updated, skeleton cut



Auto Nest

This activates the automatic nesting procedure. Supply the appropriate settings as requested in the Choose Nesting Parameters dialog box, the application will automatically position parts within the boundaries of the material shown on the screen. If you are not satisfied with the results it is possible to select all objects and reactivate the Nest function to give a different result. You may also use the select, move and rotate functions to manually alter the nest.

Note: Grouped and linked objects are nested as single parts.

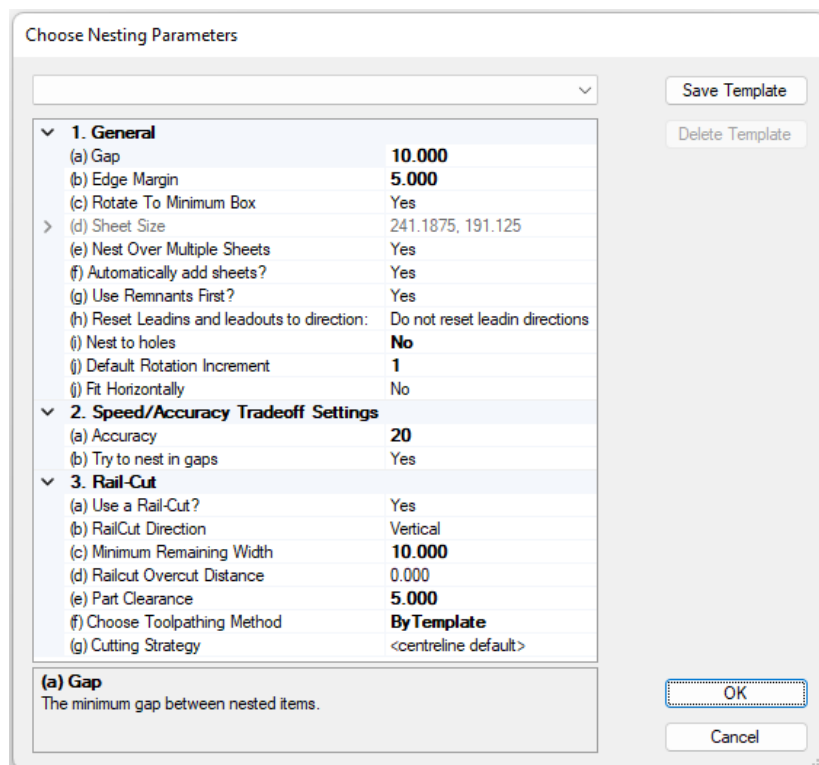


Figure: Nesting parameters dialog box.

The "Choose Nesting Parameters" options are as follows:

- **Gap.** Gap defines the minimum distance between nested items. The distance is between the edges of shapes, not the distance between toolpaths. Toolpaths are ignored during nesting.
- **Edge Margin.** Minimum allowed distance from the part to the (pre-toolpathing) edge of the material.
- **Rotate To Minimum Box.** This option will rotate nested items so that they have the smallest bounding box.
- **Sheet Size.** Shows the size of the current material.
- **Nest Over Multiple Sheets.** When sheet is full it prompts user to create a new sheet. Selecting Yes will create a new sheet with same material and size as the first sheet. Then Nest will continue for the remaining shapes.
- **Automatically Add Sheets.** This option is available when "Nest Over Multiple Sheets" is selected. Selecting Yes will automatically create a new sheet with same material and size when sheet is full. However, if this material has run out of stock then the next best material is automatically selected.

The order, in which ToolShop selects the next available material when "Automatically Add Sheet option" is selected, is as follows:

- 1) Same material and thickness from remnant (If "Use Remnants First" option is ON)
- 2) Same material and thickness with Default Size = TRUE.
- 3) Same material and thickness with Default Size = FALSE

Note: The "Automatically Add Sheets" option is not available when "Choose Nesting Parameters" dialog box is accessed via "Nesting Templates..." button on the ATP File Open window. The ATP process uses its own material handling.

- **Use Remnants First.** If the Material Library has any remnants in stock with matching material and thickness, the Nesting will proceed to use ("reserve") them first before full size sheets. The reserved materials are not removed from the stock until the sheets are exported to NC.
- **Reset Lead-Ins and Lead-Outs To Direction.** Selecting Yes will automatically re-position start points after nesting. The available options are:
 - Do not reset lead-in directions
 - Move To Top, Move To Bottom, Move To Right, Move To Left
 - Move To Top Right, Move To Top Left, Move To Bottom Right, Move To Bottom Left

With plasma cutting SE is the most advantageous.

- **Nest to holes.** Default value is **No**. If this option is selected, then nesting will try to nest smaller shapes inside holes of larger shapes.
- **Default Rotation Increment.** Default value is 1 degree, which means that the shape has 360 options to rotate. (0 degrees = No rotation)

Commented [JR12]: Added.

Commented [JR13]: Nesting should only choose from correct thickness. Removed text indicating otherwise.

- **Fit Horizontally.** Default value is **No**. This means that Nesting tries to fit all objects to the left in order to maximize the size of any offcut on the right. A half-filled sheet will have objects only on the left-hand side. The right-hand half will be empty.

If the setting **Fit Horizontal** is set to **Yes**, then nesting will instead try to fit all shapes to the bottom of the sheet in order to maximize the size of any offcut to the top. A half-filled sheet will have objects only on the bottom half. The top half will be empty. The nesting will also try to rotate nested shapes so that their longest side is facing down. This is helpful if you are plasma cutting lots of small narrow shapes and you want to prevent shapes from falling through the gaps between the slats.

What it does not mean:

Setting **Fit Horizontal Yes** does not necessarily try to arrange all individual objects horizontally, although that may likely happen. If all shapes need to be laid horizontally, this is controlled by setting the **Default Rotation Increment** value within the "Choose Nesting parameters" dialog box or by using the ROTATIONANGLE column within ATP list files (CSV, XLS, etc.). Please set this value to 180. This will allow the shapes to be rotated 180 degrees from their current orientation. When you wish to restrict the allowable rotation, it is also recommended to set the **Rotate To Minimum Box** setting to **No** when nesting.

Please Note: If the "ROTATIONANGLE" value is defined for specific DXF files within an ATP list file then that value will have priority over the "Default Rotation Increment" setting. The "Default Rotation Increment" value is used in all other instances.

- **Accuracy.** Accuracy defines how thoroughly the nesting searches for potential placement points. Note! Low values for this setting may cause the search to take longer. The default value is 20.
- **Try To Nest In Gaps.** Selecting True may lead to a 'tighter' nest
- **Use A Rail-Cut.** Selecting Yes will add a rail-cut to the nested sheet. This is usually a vertical cut.
- **Rail-Cut Direction.** The available options are: Vertical, Horizontal, LCut, TCut (vertical), TCut (horizontal) and TCut (maximum remnant size).
- **Minimum Remaining Width.** Specify the minimum desired width for any remnant. This takes into account the width of the cutting tool. If L-Cut is selected, this may become either a vertical or horizontal cut if one arm is too thin.
- **Railcut Overcut Distance.** Specify the amount of overcut. For example, the overcut of 10mm for Vertical rail-cut, would mean that both the starting point and end point are 10mm outside of the material edge.
- **Choose Toolpathing Method.**
 - ByTemplate. Selecting this option will enable the "Cutting Strategy" option.
 - Cutting Strategy. Choose the cutting strategy for the rail-cut. Selecting "<centreline default>" value will try to use previously used centreline template, if available. The rest of the options in this list depend on the currently selected machine configuration and whether user has saved any toolpathing templates.
 - ByLayer. Selecting this option will enable the "Layer" option.
 - Layer. Choose the layer which you want to use to generate the rail-cut. If the selected layer does not have a valid toolpathing template linked to it, a warning message is shown.

Note: You can link individual layers to toolpathing templates in Layer Manager.

- **Cutting Strategy or Layer.** This option depends on the chosen Toolpathing Method.
- **Save Template.** Will save your preferred nesting parameters as a template.
- **Delete Template.** Will delete the selected template from the template list.

If you are nesting with "Nest over multiple sheets" option enabled and shapes don't fit on the current sheet, a dialog is shown prompting you to create a new sheet.

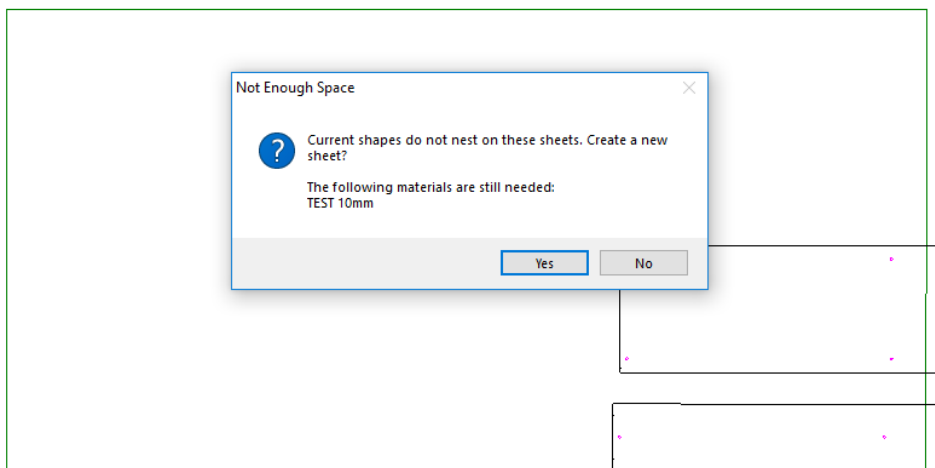


Figure: Nesting over multiple sheets. If "Automatically Add Sheets" is No then user is asked whether nesting will continue on a new sheet.

If you click Yes button ToolShop program shows you another dialog where you can define the dimensions of the new sheet.

If you click No button, the Nesting is stopped and shapes that do not fit on the current sheet are selected as default. You can then drag these shapes outside of the sheet or delete them.

You can cancel the nesting procedure anytime.



Manual Nest

Click this button to enable user interacted nesting. Define nesting gap and edge margin to the provided textboxes. Nesting gap is the distance between the edge of the shape and the next closest edge of a shape. Edge margin is the distance between sheet edge and nested shapes. By using the mouse you can move and rotate shapes on the sheet. Also you can delete shapes from the sheet.

<input checked="" type="checkbox"/>	Gap	10
Manual Nest	Edge Margin	5

Figure: Type in desired nesting gap and edge margin to the provided textboxes.

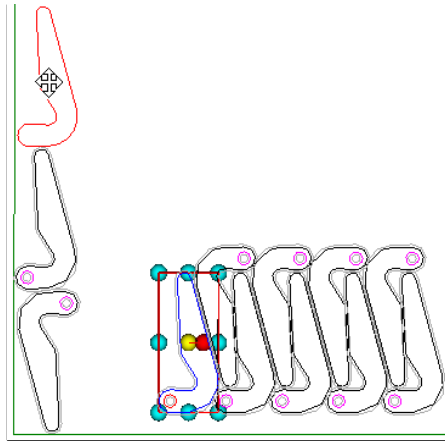


Figure: Manually nesting the shapes.

Nest To Direction

These options can be used to assist you in manual nesting. The available options are:



Nest To Left

Move selection left as far as possible.



Nest To Right

Move selection right as far as possible.



Nest To Bottom

Move selection down as far as possible.



Nest To Top

Move selection up as far as possible.



Add Rail-Cut

This function opens the “Add Rail-Cut” dialog box which allows user to add a rail-cut to the nested sheet.

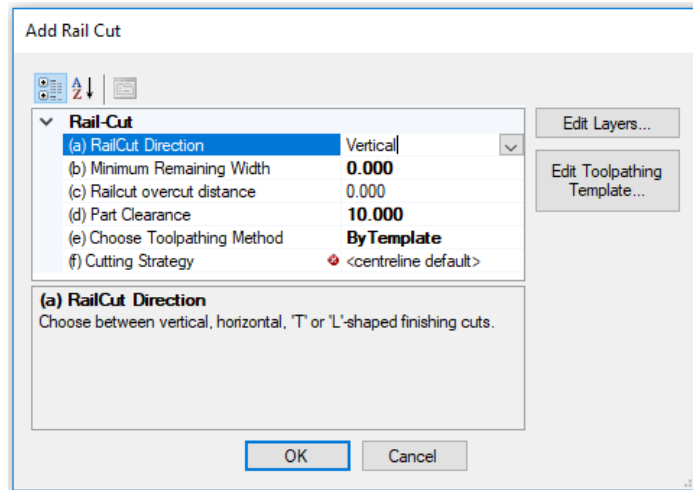


Figure: User is adding a vertical rail-cut to the nested sheet.

The “Add Rail-Cut” options are as follows:

- **Rail-Cut Direction.** The available options are: Vertical, Horizontal, LCut, TCut (vertical), TCut (horizontal) and TCut (maximum remnant size).
- **Minimum Remaining Width.** Specify the minimum desired width for any remnant. This takes into account the width of the cutting tool. If L-Cut is selected, this may become either a vertical or horizontal cut if one arm is too thin.
- **Railcut Overcut Distance.** Specify the amount of overcut. For example, the overcut of 10mm for Vertical rail-cut, would mean that both the starting point and end point are 10mm outside of the material edge.
- **Part Clearance.** Specify the minimum desired gap between shapes as a guide for how far the rail cut must be from the objects.
- **Choose Toolpathing Method.**
 - ByTemplate. Selecting this option will enable the “Cutting Strategy” option.
 - Cutting Strategy. Choose the cutting strategy for the rail-cut. Selecting “<centreline default>” value will try to use previously used centreline template, if available. The rest of the options in this list depend on the currently selected machine configuration and whether user has saved any toolpathing templates.
 - ByLayer. Selecting this option will enable the “Layer” option.
 - Layer. Choose the layer which you want to use to generate the rail-cut. If the selected layer does not have a valid toolpathing template linked to it, a warning message is shown.

Note: You can link individual layers to toolpathing templates in Layer Manager

- **Cutting Strategy or Layer.** This option depends on the chosen Toolpathing Method.
- **Edit Layers...** This option opens the Layer Manager which allows you to link individual layers to toolpathing templates.
- **Edit Toolpathing Template...** This option allows you to edit the parameters of the specific toolpathing template.



Remove Rail-Cut

The "Remove Rail-Cut" option is only available if there is a rail-cut on the current sheet. Clicking this button will remove the rail-cut.



Add Skeleton Cut

The function opens the "Skeleton Cut" dialog box which allows you to add a skeleton cut to the current work sheet. Its purpose is to cut up the skeleton of the material into more manageable sizes.

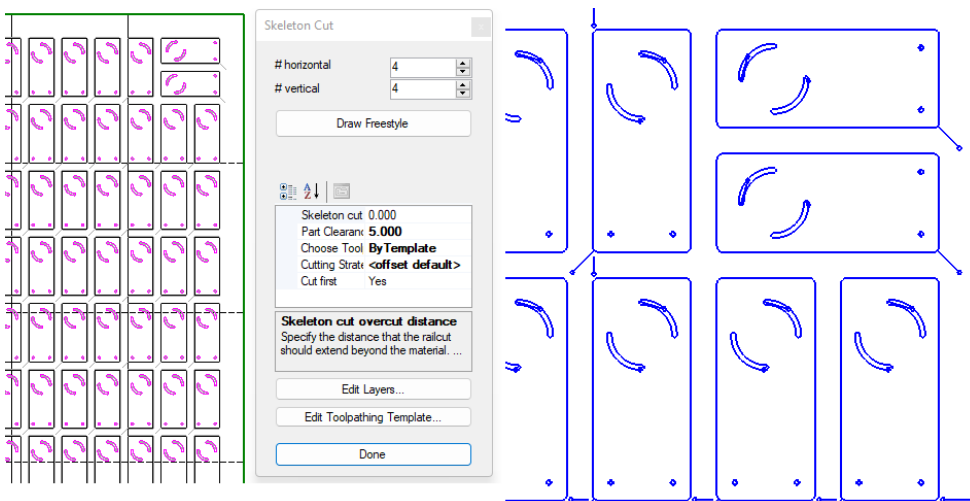


Figure: User has added both horizontal and vertical skeleton cuts.
The zoomed view of the cut file is shown.

The "Skeleton Cut" options are as follows:

- **#horizontal.** Number of skeleton cuts in X axis direction.
- **#vertical.** Number of skeleton cuts in Y axis direction. Thinner materials might only need vertical cuts.
- **Draw freestyle.** Allows you to click with mouse to add connected skeleton cut segments into the sheet. When you are finished adding the skeleton cut segments, click the "Done" button. This finalizes the skeleton cut on the sheet. Clicking "Cancel"

Freestyle Drawing" will cancel this operation and will clear previously added skeleton cut line segments.

- **Skeleton cut overcut distance.** Specify the distance that the skeleton cut should extend beyond the material. Negative values are allowed. Default value is 0.
- **Part clearance.** Specify the minimum clearance required between the edge of the shapes and the side of the skeleton cut tool.
- **Choose Toolpathing Method.**
 - ByTemplate. Selecting this option will enable the "Cutting Strategy" option.
 - Cutting Strategy. Choose the cutting strategy for the skeleton cut. Selecting "<offset default>" or "<centreline default>" value will try to use previously used offset or centreline template, if available. The rest of the options in this list depend on the currently selected machine configuration and whether user has saved any toolpathing templates.

Note: You can edit the templates by clicking on the **"Edit Toolpathing Template"** button.

- ByLayer. Selecting this option will enable the "Layer" option.
 - Layer. Choose the layer which you want to use to generate the skeleton cut. If the selected layer does not have a valid toolpathing template linked to it, a warning message is shown.

Note: You can link individual layers to toolpathing templates in Layer Manager, which you can access by clicking **"Edit Layers"** button.

- **Cut first.** The default value is Yes. The skeleton cut is done before the main cut.

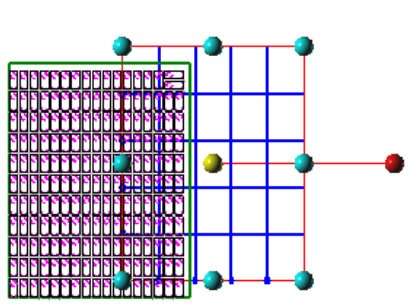


Figure: You can remove existing skeleton cut by selecting it with mouse and then pressing the Del key on the keyboard.

If you wish to remove the skeleton cut, simply select the skeleton cut by clicking it with the mouse. This highlights the skeleton cut as active. Then press the "Del" key on the keyboard.

Commented [JR14]: added



Cut Order | Manual

Click on the "Manual" cut order button to enable this mode. Then click the mouse on the first object to be cut and then click or drag mouse around the screen in the general order that you wish to process the parts. Right mouse click will show you a pop-up menu which enables you to Finish Cut Order or Cancel it. Alternatively you can re-click on the "Manual" cut order button

to process the cut order. This will also exit the cut order mode. Cut ordering is a critical function as cutting away parts which link the rest of the job in place may cause instability and movement. Cut ordering is based on the position of the start points rather than on the centre points of shapes. Inner toolpaths will be ordered before their surrounding toolpaths if the Combine function has been applied to them.

Note: At times, a shape may have been negated, and then grouped with an outer shape and so appear to be combined, but will be treated as separate shapes for the purpose of cut ordering.

Finish Cut Order

Will process the user defined cut order path and shows the cut order numbers.

Cancel

Cancels the cut ordering and switches ToolShop back to the main Mouse Selection mode.

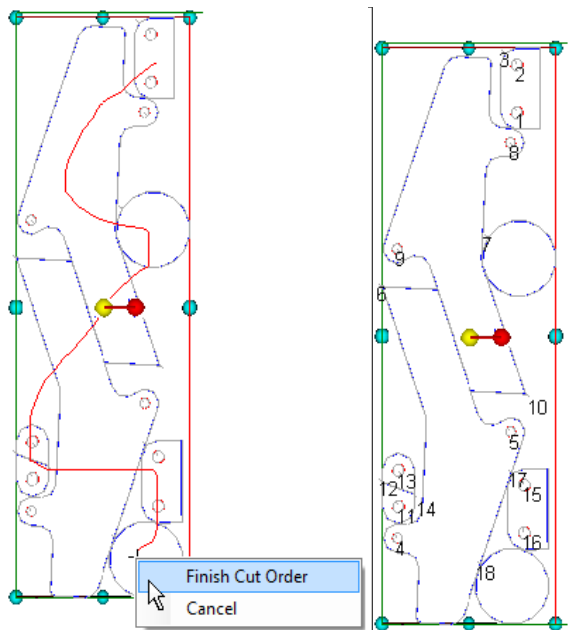


Figure: User defined cut order path. Right mouse click and selecting "Finish Cut Order" menu option will process the user defined cut order path and shows the cut order numbers.



Cut Order | Columns

This function divides the job up into a set number of columns and will automatically order the parts up and down within these columns based on proximity of the start points to the column.

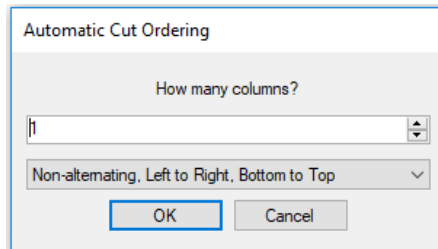
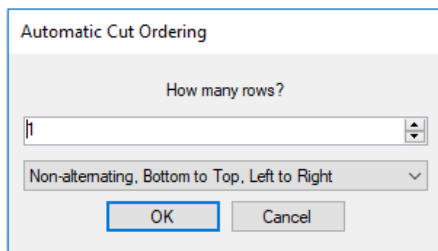


Figure: Automatic Cut Ordering dialog box when using the Columns option.



Cut Order | Rows

Ordering by rows will divide the job up into a set number of rows running horizontally across the job. The parts will be ordered across in a zigzag fashion according to their proximity to these lines.



Pic up

Figure: Automatic Cut Ordering dialog box when using the Rows option.



Cut Order | Tweak

This will open the floating Tweak Cut Order window. The window will remain open until user closes it or executes another ToolShop function. This allows the user to manually alter the cut order on the current sheet and at the same time observe the changes. The window is divided into two tab pages: **Layers** and **Tweak Cut Order**.

The **Layers** tab page shows the toolpathed layers of the current sheet. Select the layers of which cut order you wish to tweak.

Commented [JR15]: modified

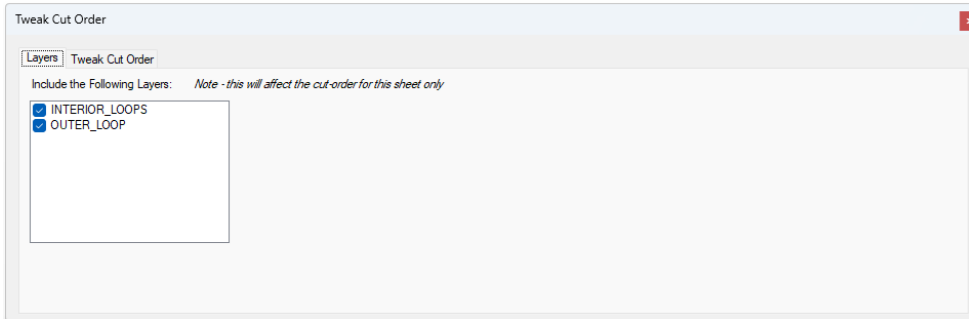


Figure: Layers that are included in the cut order tweaking.

The **Tweak Cut Order** tab page can be used to alter the cut order position of individual shapes. The changes in cut order are updated to the display in real time.

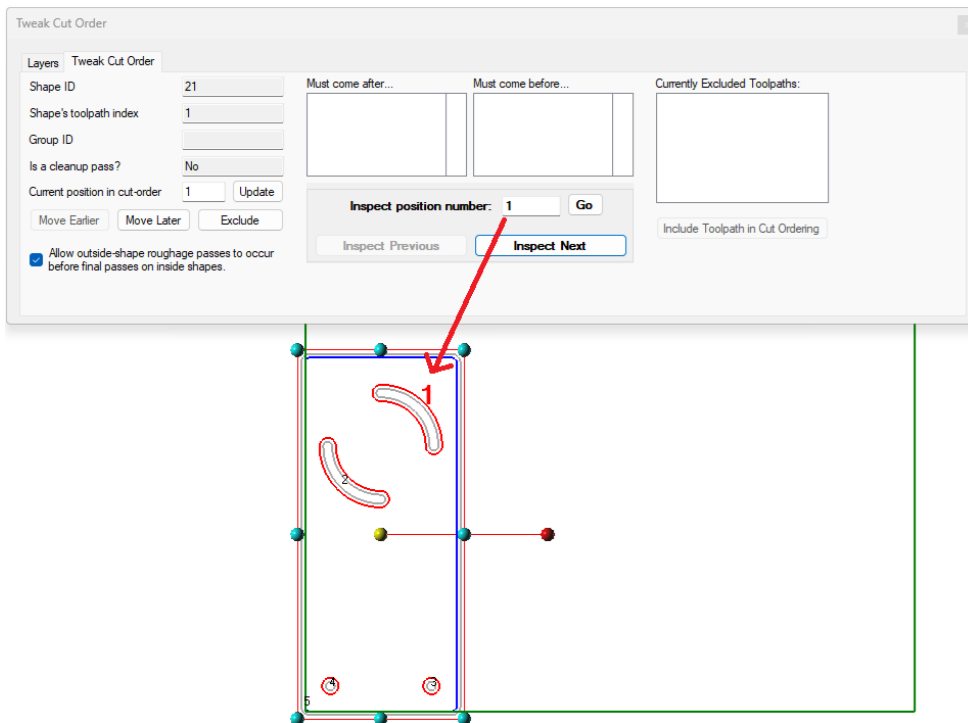


Figure: The active cut order position appears highlighted on the screen.

Commented [JR16]: modified

To modify a cut order position

- First enter into the "Inspect position number" field the cut order number you wish to go to and then click the "Go" button. Alternatively, you can use the "Inspect Previous" and "Inspect Next" buttons to navigate to the desired cut order position.

- Then type into the "Current position in cut-order" field the new desired value and click the "Update" button or use the "Move Earlier", "Move Later" and "Exclude" buttons to get the desired cut ordering. Excluded toolpaths can be added back to cut ordering by clicking the "Include Toolpath in Cut Ordering" button.

You can close this window by clicking the X button on top right corner.



Cut Order | Cut Order Settings

Commented [JR17]: added

The **Cut Order Settings** window shows the current cut order preference setting. This setting has a direct effect on all the cut ordering functions (e.g Manual/Columns/Rows). By default, the "Override cut order" option is off. This setting can also be accessed from "Export" window via "File | Output to Machine" menu. The "Override cut order" option can be used to define the order in which tools are used or shapes are cut. Use "Up" and "Down" buttons to change your **Force all first**, **Cut Order preference**, **Tool order**, **Toolpath Type order**, and the **Layer order**. Only the Cut order preference items that appear in bold text have effect on the cut ordering. The higher the "Cut Ordering" item is on the list the less other preferences can affect ordering. The **Toolpath Type order** list shows the default processing order of toolpaths for each tool. Usually there is no need to alter this setting. For example, if same plasma tool is used for both marking and cutting, then it is important that marking is done before cutting. The **Force all first** is off by default. It can be used to force the processing of selected toolpath types on the current work sheet before any other cuts. This can be especially useful with marking and drilling. The "Cut order preference" setting has no effect on the toolpath types that are processed by "Force all first".

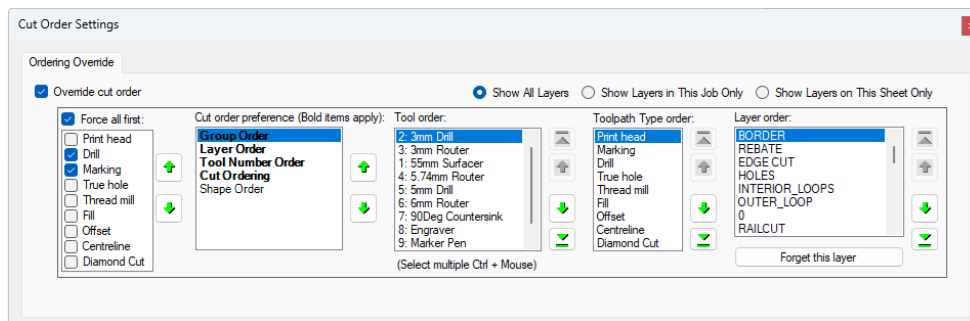


Figure: Ordering Override tab page shows the current cut order preference setting.

Commented [JR18]: modified



Printing menu

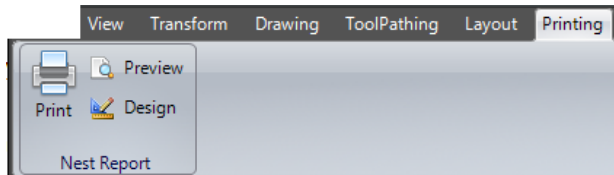


Figure: The Printing menu allows user to print a Nest Report.

By using the Printing menu you can design the layout of the printed document, preview the document and then send it to the printer. The available print documents are: nest report (part labels, nest Labels and sheet labels come with the optional ATP module). For more information please see the chapter titled “ATP Module”

ToolShop uses a similar designer for all print documents. The difference is that each of the designers may have slight variations in the items available, as well as variations in the way they're output. To learn the basics of using the designer, read the section “Using the Designer” in “An Introduction to Reporting and Labeling in ART ToolShop”.

When you open any of the print document designers they will start with current designer template. If no current designer template is defined or the default designer template is missing for the print document then user is prompted to create a new template.



Nest Report | Design

This function opens the Nest Report Designer window which allows user to design a nest report template. Upon entering the designer user is prompted to Select Printer, if required.

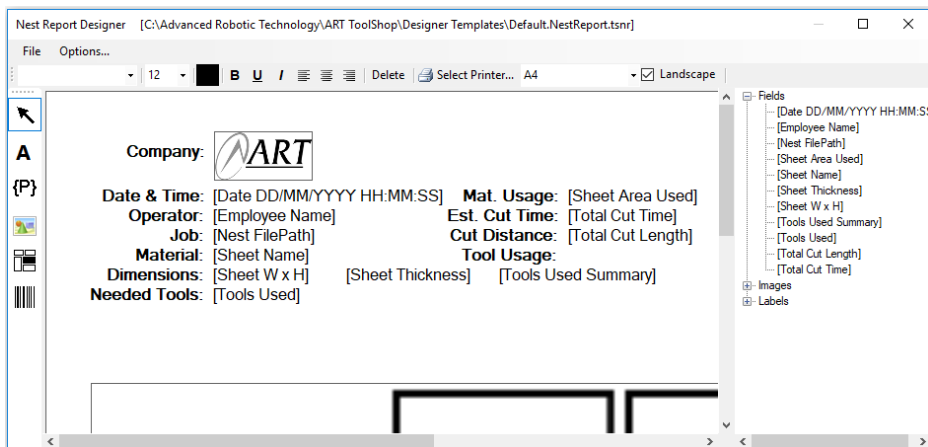


Figure: User has opened the default nest report template "Designer Templates \ Default.NestReport.tsnr".

Symbols

Below is a description of the special symbols available on a Nest Report.



Nest

Place a nest symbol onto the designer to show a representation of the nested shapes on the print out.



Figure: Representation of nested shapes on the print out.

{P} Fields

The Nest Report has a number of fields available that are specific to the Nest Report.

[Tools Used] – Provides a simple multiline list, containing one tool number and name per line.

6: 6mm Drill
7: 9mm Drill

[Tools Used Summary] – Generates a table at the upper left corner of this field. The table contains a list of all the tools used, followed by values such as: Tool name, number of starts, estimated time cutting, cut distance, time spent traveling between shapes and total distance of travel between shapes.

Tool	Starts	Cut Time	Cut Distance	Travel Time	Travel Distance
6mm Router	67	9.13min	1438.2131"	1.75min	1651.0046"

[Total Cut Length] – The total distance the tool will be cutting. This field will use the same measurement (mm, inches) unit as the ToolShop interface.

[Total Cut Time] – The estimated time the tool will spend cutting.

[Sheet Area Used] – Outputs a number representing the total area covered by parts on this nest. This is followed by the percentage of the actual area used.

[Sheet Area Used sq in] – Outputs a number representing the total area covered by parts on this nest in square inches. This is followed by the percentage of the actual area used.

[Sheet Area Used sq ft] – Outputs a number representing the total area covered by parts on this nest in square feet. This is followed by the percentage of the actual area used.

[Sheet Number] – Prints the index of this sheet in relation to the job file e.g. Sheet 1, 2 or 3.

[Is Remnant?] – This prints either 'Yes' or 'No' depending on whether the target sheet is a remnant.

[Nest Filename] – This is the filename of the ToolShop nest file. E.g. "Example_Nest.ats"

[Nest FilePath] – This outputs the entire file path to the ToolShop nest file.
E.g. "C:\WorkFiles\ToolShopNests\Example_Nest.ats"



Nest Report | Preview

This function opens the "Nest Report Preview" window which enables user to preview the nest report before printing. Click "Print" button on the preview window to send the document directly to the default printer. To change the margins click on the "Page Setup" button. To exit the preview window click "Close" or the "X" on the top right corner.

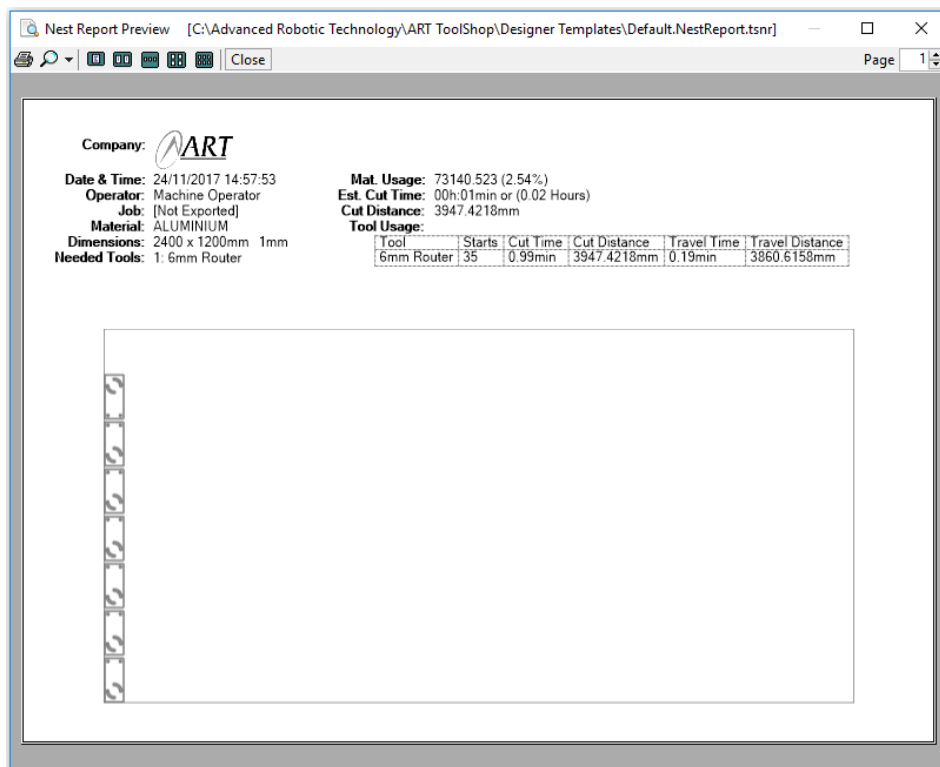


Figure: Preview of the current nest report "Designer Templates \ NestReport.tsnr".



Nest Report | Print

This function opens the "Print" dialog box which allows user to print the nest report. Select the appropriate printer and settings and click the "Print" button.

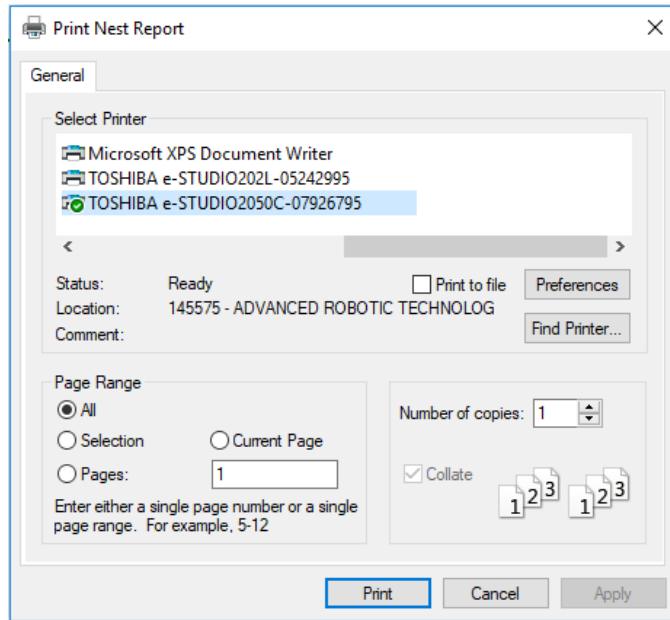


Figure: After clicking “Nest Report | Print”, user is prompted to verify printer and settings.

Select Printer

The "Select Printer" dialog box allows you to select the default printer for a particular reporting template. This dialog box is automatically shown to the user upon entering the printing template designer, if there is no default printer selected. You can also change this default printer anytime by clicking on the "Select Printer..." button in the printing template designer.

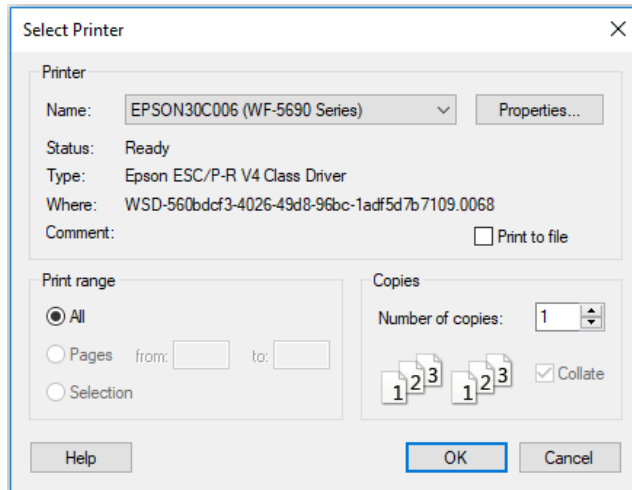


Figure: Upon entering the printing template designer user is prompted to select a default printer, if required.

Select Printing Range

The "Select Printing Range" dialog box allows you to select the printing range for a particular reporting template.

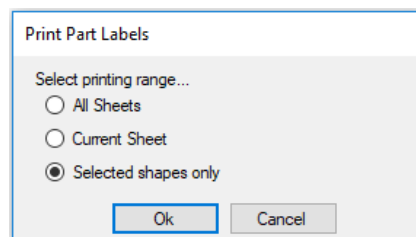


Figure: Selecting printing range when printing part labels.

Options menu

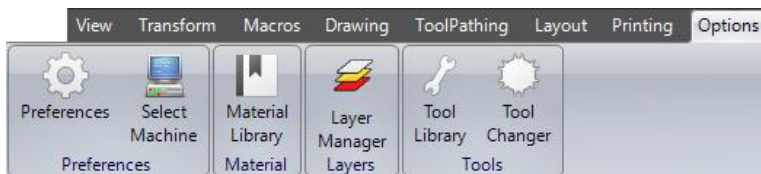


Figure: The Options menu gives access to ToolShop preferences, material and layer libraries and tool configuration.



Preferences

Specify global settings.

General Settings

- **User Interface Units.** Available units are Millimetres, Centimetres, Metres, Microinches, Inches and Feet. Click "Change Now" button to activate the new units immediately.
- **User Interface Decimal Places.** Select the number of decimal places that are shown for numerical values.
- **Travelling Height.** Machine travelling height. Default value is 5.
- **Plunge Start Height.** The default value is 1mm. Currently all toolpaths start at 1mm above surface - this setting will allow you to adjust this higher or lower. Some materials are uneven and if it is more than 1mm higher than expected the cutter can strike the surface at rapid speed causing it to break.
- **Travelling Speed.** The maximum XY feed rate while not cutting. Default value is 24000.
- **Show detailed dialogs on tool settings changes.** Enabling this option gives you greater control over tool settings changes. For advanced users only.
- **Inverse wheel zoom direction.** Inverse the mouse wheel zooming direction.
- **Enable smooth camera motion.** Default value is "Yes".
- **Anchor cursor while zooming.** Zooms to the point under mouse cursor.
- **Default Gap for Duplicate (Ctrl+D).** Default value is 5mm.
- **Nudge Distance (Arrow keys).** Default value is 5mm.
- **Nudge Rotation (Alt + Left | Right Arrow Keys).** Default value is 1 degree.
- **Use hardware acceleration.** By default, this is selected. Unselecting this setting will make ToolShop use software rendering which is slightly slower but more compatible with graphics cards. For instance, when using USB-C for the dual monitor, this setting should be unselected. Using monitor via USB-C does not support hardware acceleration. Otherwise, you can use the HDMI port for the second monitor.

- Use legacy bridge marker. Not selected.

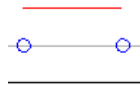


Figure: Legacy bridge marker



Figure: Default bridge marker

- Use legacy start point markers. Not selected.



Figure: Legacy start point marker



Figure: Default start point marker

Commented [JR19]: added

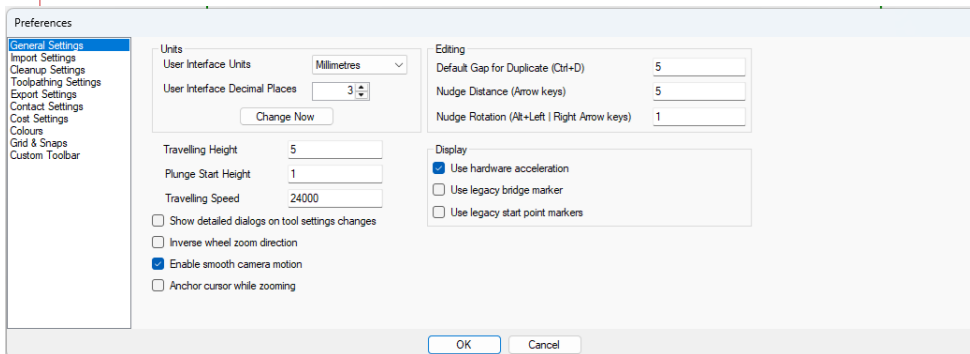


Figure: The General settings tab page.

Commented [JR20]: Pic updated

Import Settings

- **Cleanup on import.** Performs cleanup function automatically on DXF import.
- **Combine on import.** Performs combine function automatically on DXF import.
- **Smart Group on import.** Performs smart group function on DXF import.
- **Import DXF files as 2D only.** Ignores Z coordinates on DXF import. This is especially useful when DXF file has been exported from a 3D design program. Sometimes these programs export DXF files with Z coordinate that is not exactly zero.

Commented [JR21]: Added smartgrp setting

- **Reposition DXF files.** Repositions DXF drawing to X, Y coordinates 0,0 on import.
- **Ignore layers.** If this setting is ON then all shapes from DXF file(s) will be imported on layer 0. Default value is OFF which means that ToolShop will use the layers from the DXF file.
- **DXF file units.** Select the measurement unit that you used to create your DXF drawing. This setting affects the scaling of the drawing on DXF import. Default value is mm.

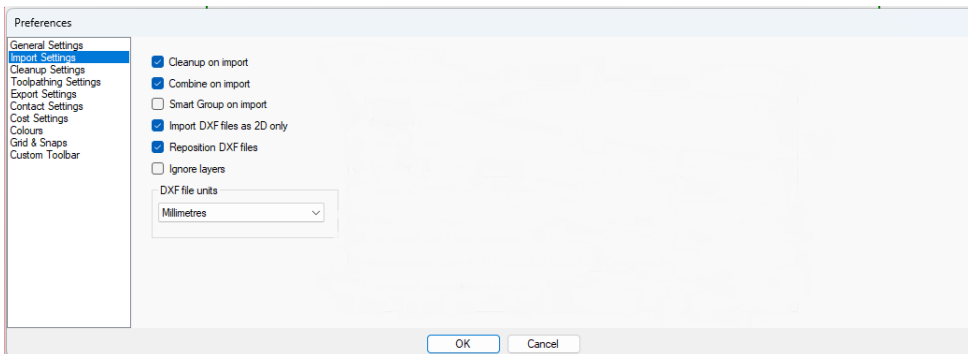


Figure: The Import settings tab page.

Commented [JR22]: Pic updated, smart group

Note: Some additional settings are available with the optional Automatic Toolpath (ATP) module. For more information please see the chapter titled "ATP Module"

Cleanup Settings

- **Auto join tolerance.** Defines the tolerance for cleanup. Default value is 0.001. The recommended value range is between 0.001 - 0.1. Greater values than 0.1 may result in distorted geometry.
- **Arc Cleanup Tolerance.** Default value is 0.1
- **Text Shape Accuracy.** Default value is 0.01. This accuracy value is used to generate text objects. The smaller the value the more accurate the rendering of the text objects.
- **Check for drawing errors.** This is selected by default. The ToolShop will run a drawing error check after cleanup procedure and after functions that modify the drawing. It will then show error markers on the job if needed. It is beneficial to have this option On as it will alert the user of possible drawing errors that are not visible to naked eye. At the moment Overlap and Cross-over checks are supported.

Commented [JR23]: added

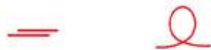


Figure: Overlapping line and Cross-over drawing error icons.

- **Overlapping lines.** Selected by default. Check for overlapping lines in the drawing.
- **Tolerance.** Default value is 1. Define the tolerance value for overlapping lines check.
- **Cross-overs.** Selected by default. Check for line segments that are crossing over other line segments.

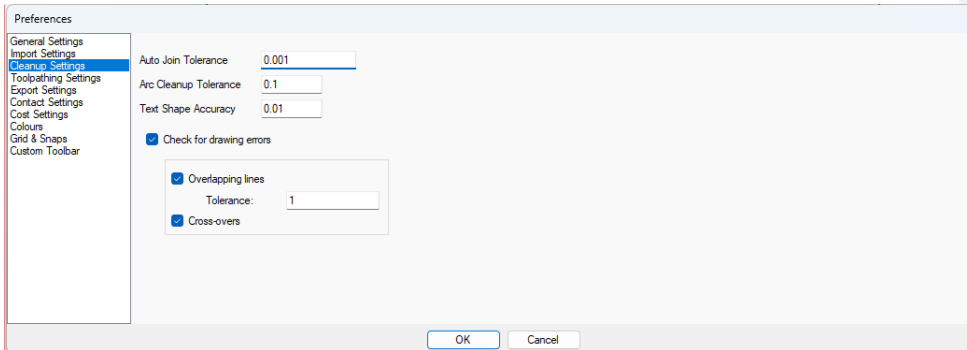


Figure: The Cleanup settings tab page.

Commented [JR24]: Pic updated, text shape accuracy

Toolpathing Settings

- **Toolpathing Tolerance.** Default value is 0.01.
- **Minimum Segment Length.** Default value is 0.1mm. This is the minimum segment length on NC output. Maximum value without affecting quality is 0.2. Very small segments, 0.1mm or smaller, may cause juddering in the machine. The issue with too many small segments is a time based issue. The machines can only process about 1000 segments a second. So, if the machine is cutting at 4000mm/min then the shortest segment that can be output is 0.0666666mm long. If the machine is cutting at 6000mm/min then it is 0.1mm. It all depends on the customer usage. Segments shorter than 1ms cut time will often cause stuttering motion.
- **Automatically move leadins to prevent collisions.** This is selected by default. When toolpathing shapes ToolShop will check that leadins won't cut into actual shapes by moving or shortening them if necessary. You may deselect this option if ToolShop appears to respond slowly when handling very large jobs.
- **Drill circular shapes only.** This is selected by default. This option is useful when you wish to quickly add drill toolpaths to all circular shapes in the job. "Select All" shapes then apply Drill toolpath using the correct diameter drill tool. Only circular shapes with the right diameter will get the drill toolpath. If you deselect this option then all selected shapes will get drill toolpaths in their centre coordinates.
- **Retract above surface.** If this option is selected then "Peck Retract Distance" (for drills) and "Default Redraw Lift Height for router bits" are absolute values from surface, otherwise they are offset values from tool's current depth.
- **Default Redraw Lift Height for router bits.** Default value is 16. This is the default redraw lift height when using a router bit for drilling.
- **Display a warning if any of the selected shape(s) fail to toolpath.** A warning message is shown to the user if any of the selected shapes failed to toolpath.
- **Select and Zoom in to the shape(s) that failed to toolpath.** (As default this is Off e.g only warning is shown). If selected TS selects the shapes that failed to toolpath and zooms in to them and then shows the warning.

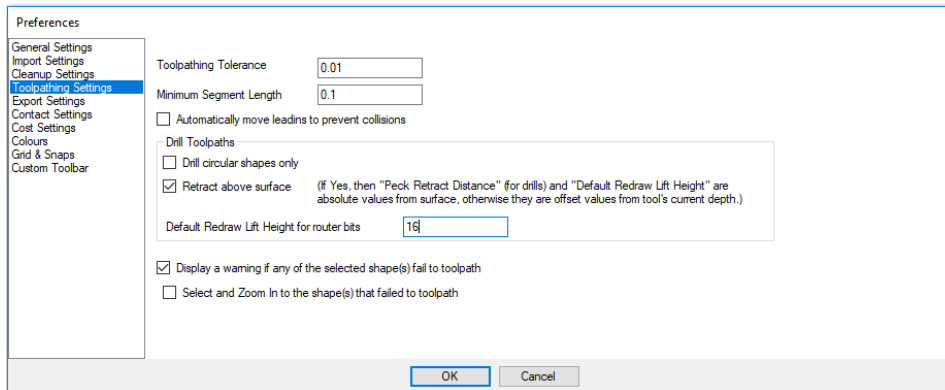


Figure: The Toolpathing settings tab page.

Export Settings

Choose the default settings for NC output.

- **Override cut order.** Choose whether the “Override cut order” option is ticked on the Export dialog box as default. Use “Up” and “Down” buttons to change your cut order preference, tool order and toolpath type order.
- **Force All First.** The “Force All First” option forces selected toolpath types to be cut first. You can alter the “Force All First” processing order, if necessary, e.g. you have more than one option selected. For example, you may wish to do the marking or drilling for the whole sheet first before proceeding cutting the individual part. After the “Force All First” toolpaths are processed the remaining toolpaths are processed using the usual cut order override settings.
- **Specific layers.** Choose whether the “Specific layers” option is ticked on the Export dialog box as default.
- **Add Material Label (DXF Only).** Adds material information text field into DXF files exported from the ToolShop.

Commented [JR25]: Added

Commented [JR26]: Added

Commented [JR27]: added

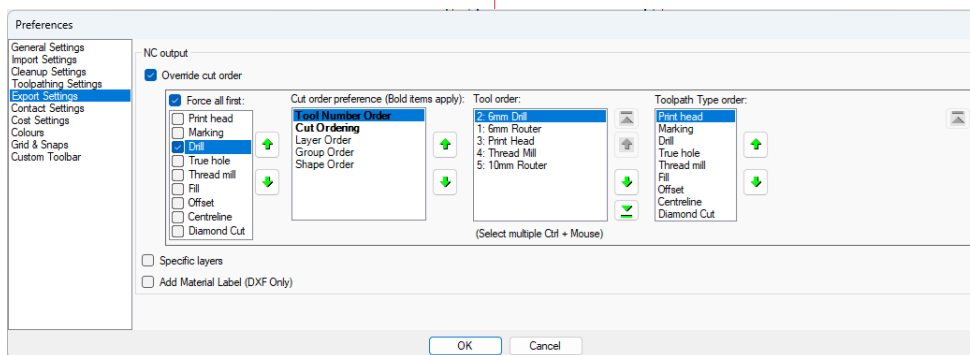


Figure: The Export settings tab page.

Commented [JR28]: Pic updated, force all first



Contact Settings

Contact information fields can be used in Nest Report and Label printouts.

- **Business Name.**
- **Employee Name.**
- **Phone Number.**
- **Fax Number.**
- **Email Address.**
- **Website.**
- **Postal Address.**

Preferences	
General Settings	
Import Settings	
Cleanup Settings	
Toolpathing Settings	
Export Settings	
Contact Settings	
Cost Settings	
Colours	
Grid & Snaps	
Custom Toolbar	

Business Name:	ART Australasia Pty Ltd
Employee Name:	Machine Operator
Phone Number:	+61 7 33936555
Fax Number:	
Email Address:	
Website:	www.artcnc.com
Postal Address:	

OK Cancel

Figure: The Contact settings tab page.

Cost Settings

The values in these fields are mainly used in the nest report for estimating cut time for the job.

- **Travelling Speed.** This value can be changed via General Settings tab page.
- **Tool Change Time.** Estimated time that it takes for machine to change its tool.
- **Raising/Lowering Speed.** Input here used Raising/Lowering speed.
- **Cut Time Correction Factor.** This value can be used as a 'correction factor' to alter the estimated total cut time.

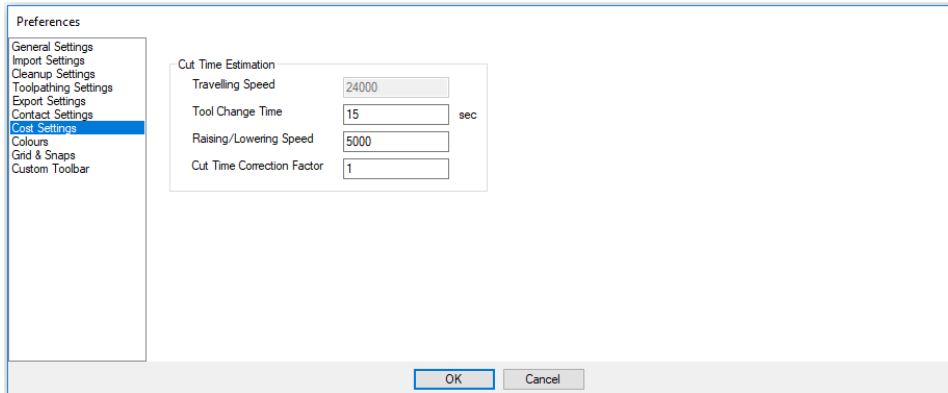


Figure: The Cost settings tab page.

Colours

You can change the default user interface colours from here. Click “Restore Defaults” button to revert back to the original colour settings.

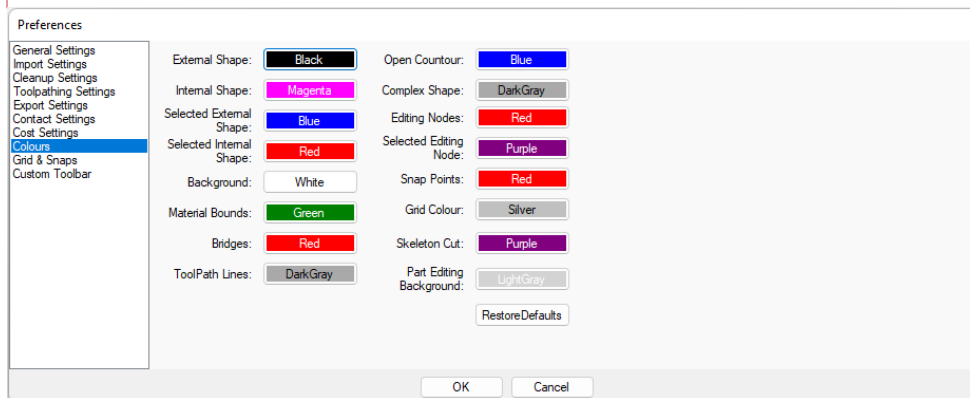


Figure: The Colours tab page.

Commented [JR29]: Pic updated

Grid & Snaps

You can change the default grid and snap settings from here. These settings affect selection, moving and drawing functions.

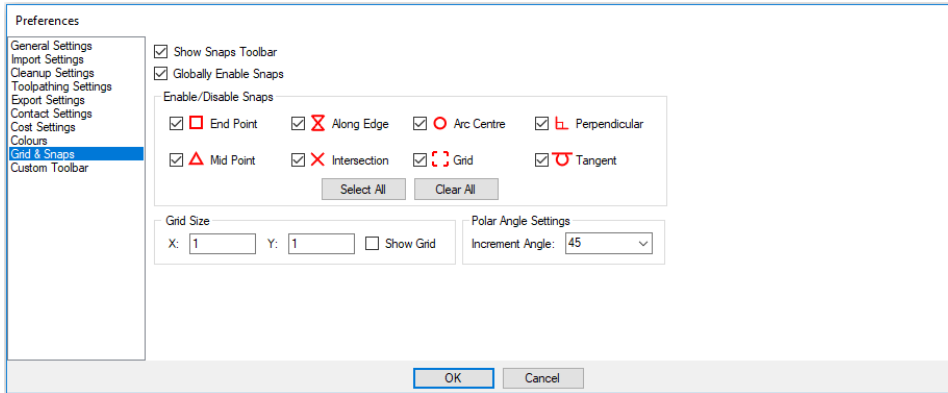


Figure: The Grid & Snaps settings tab page.

- **Show Snaps Toolbar.** Show or hide Snaps toolbar.
- **Globally Enable Snaps.** Turns all snaps On/Off without changing settings.
 - **End Point.** Snap mouse cursor to the closest end point.
 - **Along Edge.** Snaps mouse cursor to the closest point along the path.
 - **Arc Centre.** Snaps mouse cursor to the centre point of the arc or circle.
 - **Perpendicular.** Snaps mouse cursor to perpendicular points adjacent to the original reference.
 - **Mid Point.** Snaps mouse cursor to the midpoint of an arc or a line.
 - **Intersection.** Snaps mouse cursor to intersections of overlapping segments.
 - **Grid.** Snaps mouse cursor to the grid points.
 - **Tangent.** Snaps mouse cursor to a tangent point on a circle.

You can click "Select All" button to activate all snap options. Clicking "Clear All" button will clear all snap options.

- **Grid Size.** Define the size of the grid in X and Y directions. The default grid size is 1mm x 1mm.
- **Show Grid.** Select this option to show the grid. Please note that the grid appears only if you have zoomed close enough to the sheet.
- **Polar Angle Settings.** When Shift – key is held down and you make a selection or draw items, it snaps to the closest angle. Define the angle increment. The default value is 45 degrees.

Custom Toolbar

The Custom Toolbar will allow you to add most frequently used commands into a separate toolbar. Select the commands you wish to add and then click the "Add" button. You can use the "Remove" button to remove commands from the custom toolbar. You can change the order of custom toolbar buttons by using the "Up" and "Down" buttons.

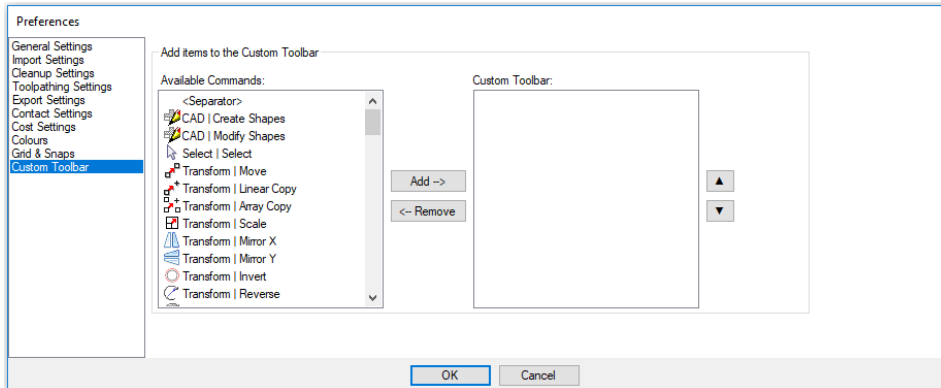


Figure: The Custom Toolbar tab page.



Select Machine

Specify the current machine configuration. Select appropriate configuration from the “Imported configurations” list and click “OK” button. This will show user a confirmation message that current definition is changed and will be backed up to the Drivers folder. Accepting the confirmation will back up the old definitions and then load the new configuration to the ToolShop program. The “Apply” button will allow you to change the current machine configuration without closing the “Select Machine” dialog box.

The configuration files have *.mach extension. These files contain the tool changer, tool library and some global settings for the specified machine.

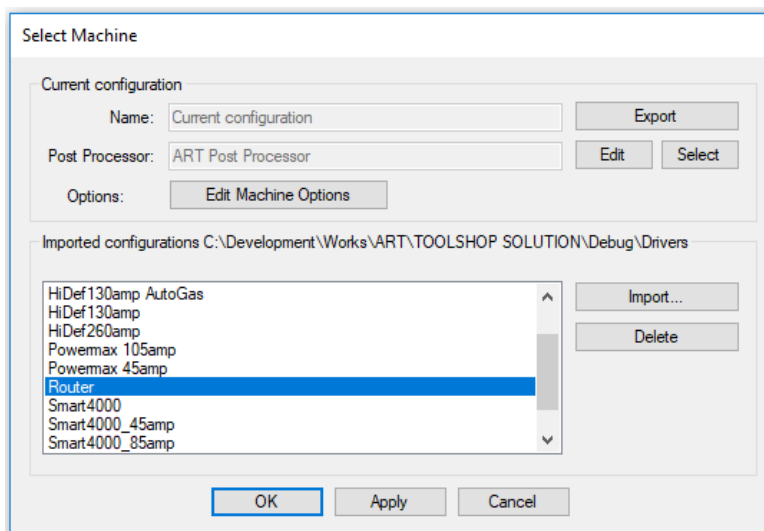


Figure: The Select Machine allows user to specify the current machine configuration

Note: If ToolShop was installed on a new PC and no previous configuration exist, then ToolShop will prompt user to import configuration files from the ToolShop installation CD during program start-up. Otherwise you can use the "Options | Select Machine" menu and use the import function to import new configurations.

The "Select Machine" options are as follows:

- **Current Configuration.**
 - **Name.** The name of the current machine configuration.
 - **Export.** The Export button allows you to export your current configuration as a text file to a portable media or network drive for backup or diagnostic purposes.
 - **Post Processor.** The name of the selected post-processor. The default post-processor is "ART Post Processor".
 - **Edit** button will open the "Post Processor Editor" dialog box which allows you to change the parameters of the current post-processor. See the "Post Processor Editor" manual for more details.
 - **Select** button will open the "Select Default Processor" dialog box which enables you to select a different post-processor file. You can create additional post-processor files by simply making another copy of the current post processor file in the \Posts folder or you can use the "Save As" button when editing the current post-processor settings.
 - **Options.**
 - **Edit Machine Options** button will open the "Edit Output Options" dialog box. This will allow you to add additional custom options/settings into the toolpathing dialog box. These values are then used by the post-processor. See the "Post Processor Editor" manual for more details.
- **Imported configurations.** This section shows currently available machine configurations.
 - **Import** button allows you to import new configurations for different machines like Smart4000, HiDef130, HiDef260 and Router etc. Just select appropriate configuration file from the "Drivers" folder from the ToolShop installation CD. Import will then copy the selected machine file to the \Drivers subfolder under ToolShop folder.
 - **Delete** button allows you to remove machine files from your "Drivers" folder.

Post Processor Editor

To access the “Post Processor Editor”, click on the “Edit” button on the “Select Machine” dialog box. This will allow you to change the parameters of the currently selected post-processor. See the “Post Processor Editor” manual for more details.

Post Processor Editor

General Machine Motion Arcs Spindle Feedrate Start/End Tool Change Events Variables Custom Variables

File
 Exported File Extension: CNC
 File Start Reading Char:
 File Stop Reading Char:

Incremental Filename
☐ Generate incremental filename
 Prefix:
 Postfix:
 Increment by: 1
 Start At: 1
 Currently At: 44
☐ End At: 999
 # Leading 0's: 3

Mode
☒ Absolute Code: G90
☐ Incremental Code: G91

Units
☐ Imperial :: Inches Code: G70
☒ Metric :: Millimeters Code: G71

Block Format
☒ None ☐ Space ☐ Tabs
☐ User Defined: ,
 End of Block Char:
☐ Always Output + Sign

Modal Output
☐ G Code ☐ Feedrate
☐ Coordinate ☒ Spindle Speed

Sequence Numbers
☐ Use Sequence Numbers
 Prefix Char: N
 Increment: 1
 Start Value: 1
☒ Show Leading Zeros
 Number of Digits: 4

Comments
☒ Output Comments
 Comment Start Char: (
 Comment Start Char:)
 Output Sequence Number: ☐

Preview Save Save As... Close

Figure: Editing Post Processor settings.

Edit Output Options

To access the “Edit Output Options” dialog box click on the “Edit Machine Options” button on the “Select Machine” dialog box. This will allow you to add additional custom options/settings into the toolpathing dialog box. These values are then used by the post-processor. See the “Post Processor Editor” manual for more details.

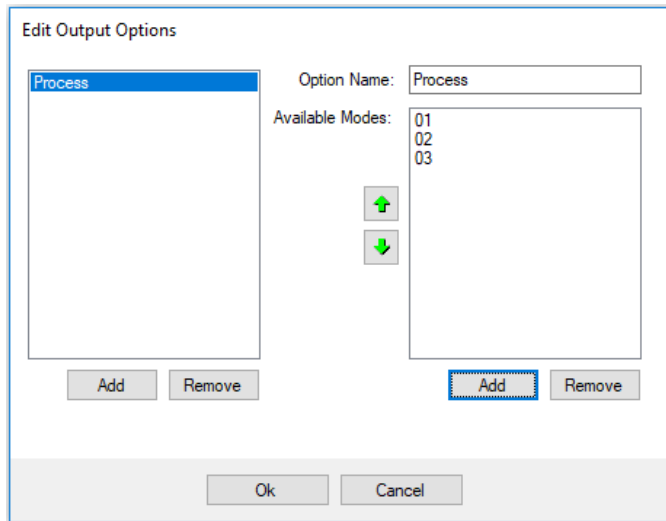


Figure: Adding custom settings via “Edit Output Options” dialog box. These settings will appear in the toolpathing dialog box.



Material library

Material library allows you to keep track of the current material stock. To enter new materials into the library, click on the blank line at the bottom of the list and type in a new material name. By highlighting any entry on the grid you may alter the settings. Materials entered via this dialog box will be available from the “View | Select Material” function.

Material Library										
Materials										
MaterialID	Name	Thickness	Width	Height	Default Size	Remnant	Infinite Quantity	Negative Quantity	Quantity	
7	MILD STEEL	1.000	2400.000	1200.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	<input type="text"/>
8	MILD STEEL	2.000	2400.000	1200.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	<input type="text"/>
9	MILD STEEL	3.000	2400.000	1200.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	<input type="text"/>
10	MILD STEEL	4.000	2400.000	1200.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	<input type="text"/>
11	MILD STEEL	5.000	2400.000	1200.000	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	<input type="text"/>

Figure: The Material Library allows the user to define new materials.

Commented [JR30]: Pic updated

The options for each material are as follows:

- **MaterialID.** Material ID number.
- **Name.** Material name.
- **Thickness.** Material thickness.
- **Width.** Material width.
- **Height.** Material height.
- **Default Size.** Sometimes you might have different sizes available on the same material and thickness. Select the material size that is the most preferable option or is simply used more often. This will affect automatic nesting and ATP functions.
- **Remnant.** You can define a piece of material as a remnant. Each remnant has a Quantity of 1. If you have enabled the Rail-cut option in the Nesting options then ToolShop will automatically add any new remnants to the Material Library on NC output. If you are using the "Use Remnants First?" Nesting option then ToolShop will first try to use any available remnants before proceeding to use fresh material stock.
- **Infinite Quantity.** The material never runs out of stock.
- **Negative Quantity.** The material stock is allowed to go negative.
- **Quantity.** Define the available material quantity.
- **Shape.** Picture of the sheet.

Note: Some additional features, such as "Print Stock Labels for Selected" and "Material Templates" are available with the optional Automatic Toolpath (ATP) module. For more information please see the chapter titled "ATP Module".



Layer Manager

This function shows the Layer Manager dialog box. It allows user to browse the layers that were imported from the DXF drawing. These layers are listed under the "Job layers" tab. You can also define temporary layers by clicking "Add layer to job" button. This will add a new row where you can enter the layer name. The temporary layers are handy when using the "Edit / Move to Layer" function. The "Job layers" list is updated each time you open or import a new drawing to reflect the layer list of the drawing. By clicking on "Delete layer from job" button you can remove existing layers. "Rename Layer" button will allow you to rename a layer. If you wish to hide a layer remove the tick off the "Visible" column. Above the "Visible" column is a tick box that allows you to select or deselect the "Visible" option for all existing layers. The ToolShop screen at the background is redrawn immediately to reflect the change. This makes it easier to distinguish between different layers. You may also want to lock certain layers to prevent any changes to them.

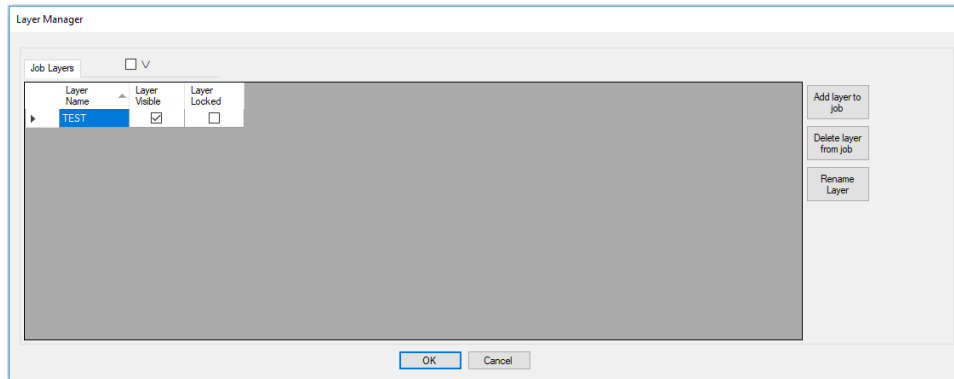


Figure: The Layer Manager allows user to browse the layers that were imported from the DXF drawing.

Note: Some additional features, such as an enhanced Layer Manager are available with the optional Automatic Toolpath (ATP) module. For more information please see the chapter titled "ATP Module".



Tool Library

The Tool Library dialog box enables you to enter tool names as well as default parameters for each tool. Create one new tool for each type of consumable used on the machine.

Note: By using "Options | Select Machine" option you can import predefined Tool libraries for different machines like Smart4000, Hidef130 and Hidef260 plasma cutters.

Note: In this version of ToolShop users can create toolpaths without linking tools to a tool number. However, before exporting a job to a machine, user will need to link the used tools into tool numbers in the tool changer function.

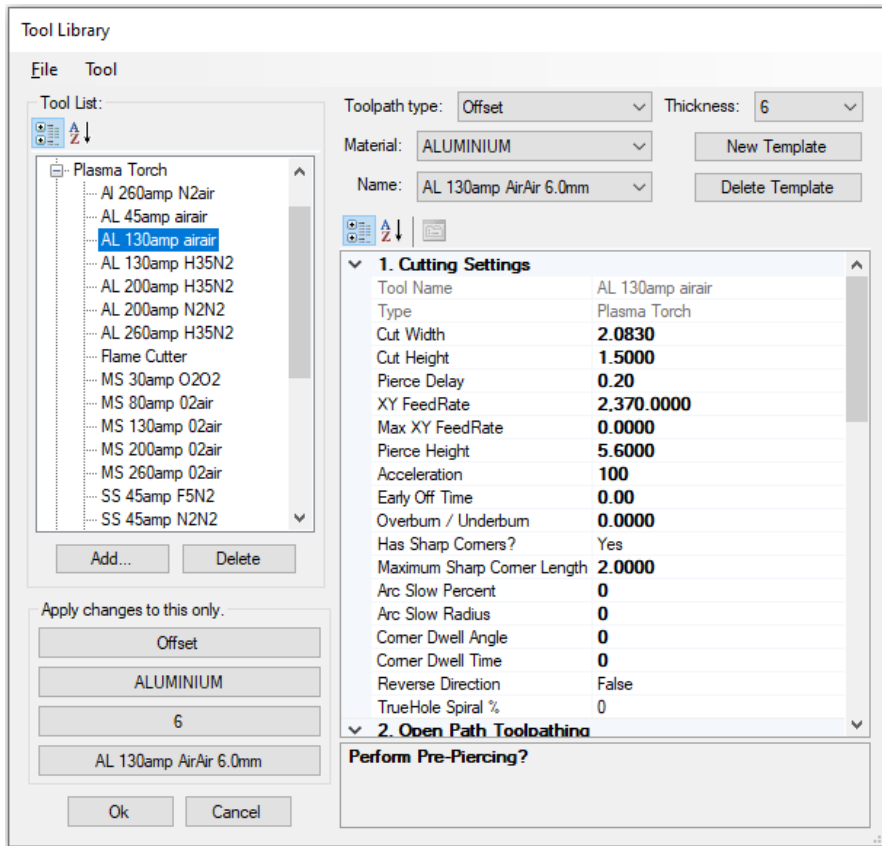


Figure: Using Tool Library dialog box you can enter tool names and their default parameters.

Adding New Tool

Click the Add button to add new tools to the Tool Library. After clicking the Add button, it will then ask user to enter the new tool name.

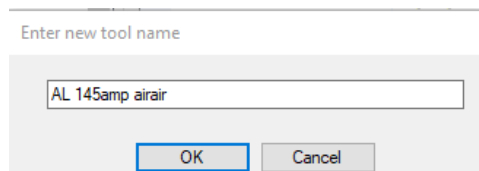


Figure: Entering a name for the new tool.

After entering the tool name, user will be asked to choose the tool type.

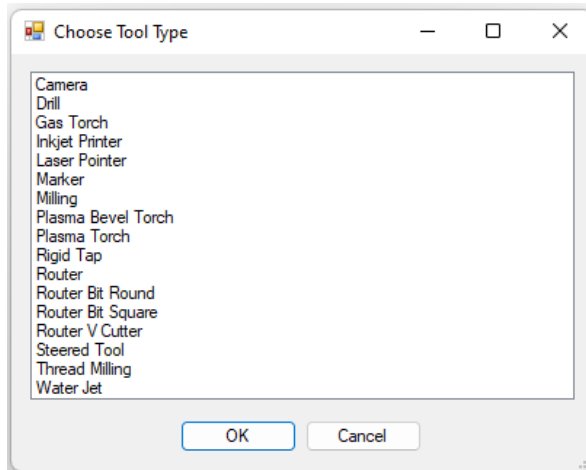


Figure: Selecting the tool type (tool category).

The tool category names (Plasma Torch, e.g.) will appear in the tool list automatically when you add at least one tool from the specified category.

Deleting Tool

Click the **Delete** button to delete the selected tool and its templates.

Adding Template

First select a tool by clicking an existing tool name. If you wish to copy properties from an existing template, then make sure you have that template selected prior to clicking the “New Template” button. Click the “New Template” button. The user is prompted to make a copy of the selected template.

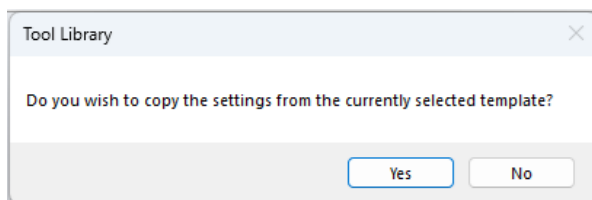


Figure: Creating a new template from an existing template.

Choose “Yes” if you wish to do so. In the “New Tool Template” window you can select the “Template Type” and whether the template should be linked to a specific material. By default, the “Apply to specific material name” and “Apply to specific thicknesses” options are not selected, e.g. the material information is not relevant. This means that the template’s material name is saved as “ALL” and the thickness value is saved as 0, e.g. it applies to “ALL”

thicknesses. The “Template Name” field has “Default” as the initial value. You may change this to your desired template name.

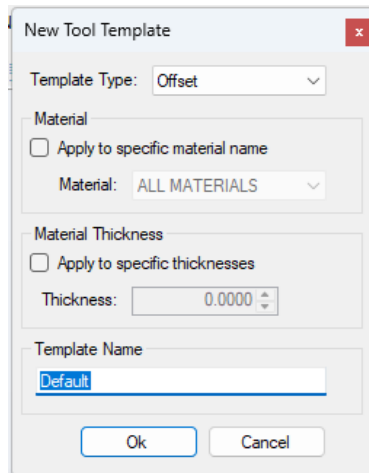


Figure: Type in the template name unless you wish to use the “Default” name.

There are some benefits in choosing to use the “Default” as the template name. The ability to have multiple “Default” templates means you don’t need to creatively invent a new name all the time. This is great if you only have one toolpath for each combination of Tool/Material/Thickness. However, you can only have one “Default” named template for each Tool/Material/Thickness combination. You can also change the range to be just the Tool/Material combination and apply it to any thickness. In this instance, select the “Apply to specific material name” option and choose the appropriate material name. Leave the “Apply to specific thicknesses” option unchecked.

Selecting Template

First select a tool by clicking an existing tool name on the Tool List. This will retrieve the first available template for that tool. All the template properties including Toolpath Type, Material, Thickness and Name are shown on the right-hand side of the window. To find a specific template, select first the correct Toolpath Type from the drop-down list. This will then populate the Material and Thickness drop-down lists according to available templates. Select the Material and the Thickness. Finally, choose the template from the “Name” drop-down list. The template properties are then shown on the right-hand side property window. Also, the “Apply changes to this only” filter buttons on the left-hand side are filled in.

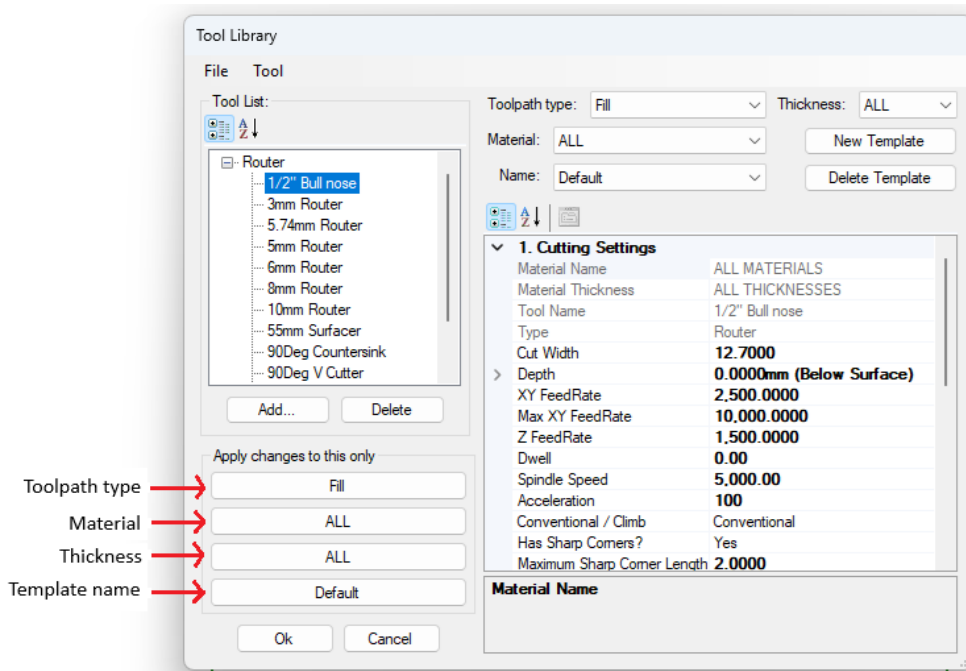


Figure: By default, changes apply to the active tool's selected template only. If you wish a range of templates to have the same properties, then click on the filter buttons to change the scope. This will need to be done before clicking the "OK" button.

Apply changes to this only filter buttons

After selecting a tool and its template, you can use the filter buttons to define the range of templates that should be updated to have these same properties. By default, these are set so that changes apply to the active tool's selected template only. As an example, we use the template from the above picture. Click the individual button to toggle between the saved value and the other value.

For example, we select "All Toolpath Types" for the 1/2" Bull nose tool.

- Toolpath type
 - Fill
 - **All Toolpath Types**
- Material
 - **ALL**
 - All Materials
- Thickness
 - **ALL**
 - All Thicknesses
- Template name
 - **Default**
 - All Names

The "Apply changes to this only" text will change to reflect how many templates are affected. In this instance, "Apply changes to... (5 matching)".

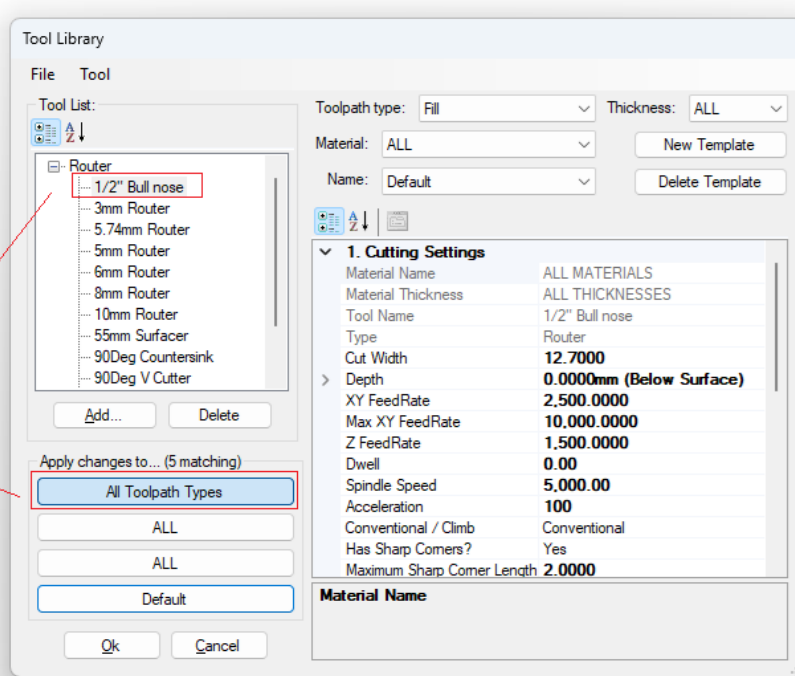


Figure: User has selected the "All Toolpath Types" option for the 1/2" Bull nose tool. When user clicks "OK" button the selected template's properties are copied to all 1/2" Bull nose tool's templates where the template name is "Default".

When you click the OK button, the changes are copied to these templates.

OK button

Click OK button to save changes into the Tool Library.

Cancel button

Exit the dialog box without saving changes.

Commented [JR31]: Updated



Tool Changer

The Tool Changer dialog box provides the link between the toolpath and the cnc output. The ToolShop must know which tool number to apply to a specific toolpath. When the machine reads the CNC file it receives a number which identifies which tool to use for this job. You must ensure that the same tools are listed under the same numbers in both the ToolShop application and the machines controller.

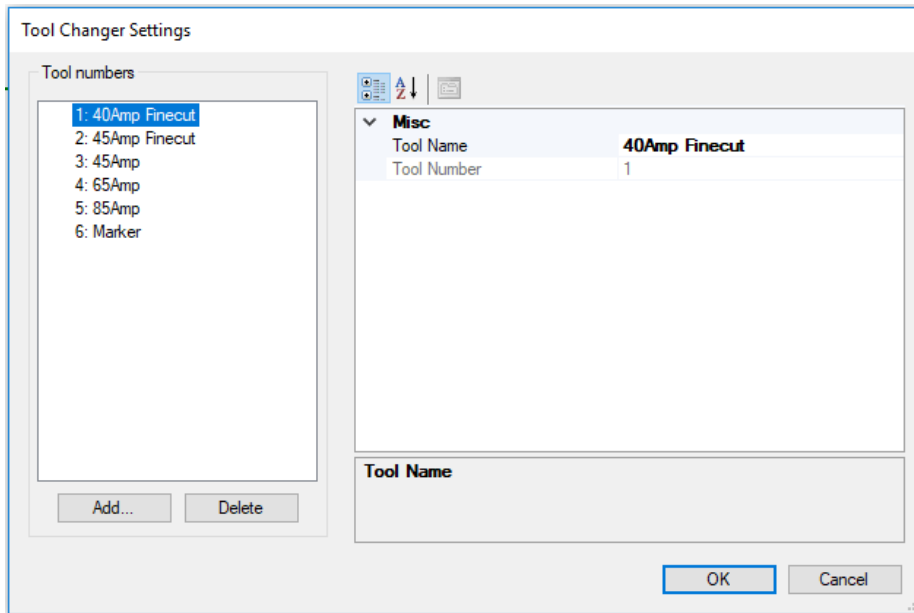


Figure: Tool Changer dialog box.

By clicking the Add button you can add new tool numbers to the list. After adding the new tool number select appropriate tool to link with the number.

By clicking the Delete button you will be able to delete the selected tool number.

Changes to these definitions are not saved until you click the Ok button. These definitions are saved into the **ToolChanger.soap** file.

Help menu

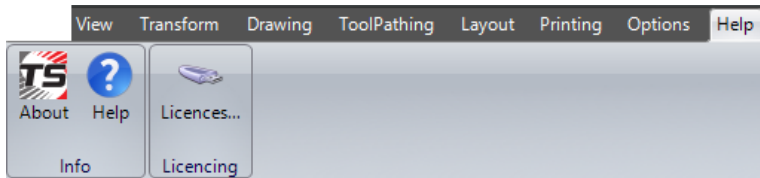


Figure: The help menu provides access to the product documentation.



About ToolShop



Figure: The About dialog box shows the ToolShop version information.



Help

This menu item will show the ToolShop user manual. You can also open the help anytime by pressing F1 key from the keyboard.



Licences

The Licences dialog box enables you to view current licensing information.

Licences

To acquire a licence file, please copy and paste the following Installation ID into your email, and then send it to toolshop@advancedrobotic.com. The support phone is : 61 7 33936555

Installation ID

3446696753663070305766762F6345736252794F7A71736669737531583934394A4753345754594D516F3362546D703536572F6E6C79787552674449516C5248346F414F716B31486D6A58484A5159315A61596149667844482F4177587A56496F416E32502B727569736234626A5464704D356338736966444637384E6B5067

Security dongle (Black)

Hardware Serial:	8037F03C	User ID:	9225136
Current Tag:	FFFFFFFF	Time:	07/17/2023 08:16:00
Current Password:	FF FF FF FF FF FF FF	<input data-bbox="762 1037 970 1064" type="button" value="Update dongle from file..."/>	

Licence file

F:\jtest2.licence

Licensed modules

ARTTOOLPATH.DLL
 TOOLSHOP.EXE
 ARTNESTING.DLL
 ARTCAD.DLL
 ARTATP.DLL

Figure: The Licences dialog box shows current licensed modules.

Commented [JR32]: Pic updated

Activating ToolShop

When you start ToolShop for the very first time it may show you this "Dongle Not Found" dialog box.

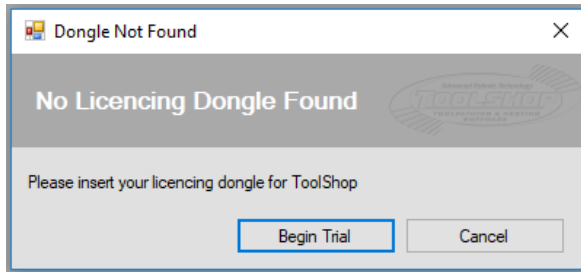


Figure: ToolShop requires a security dongle and a license file or an online license to run.

Activating with security dongle and license file

If you have a security dongle please insert it to computer's USB port. If the security dongle is not recognized for some reason, try inserting it to a USB 2.0 port. The USB 2.0 ports are of black colour and USB 3.0 ports are of blue colour. Most computers have usually at least one USB 2.0 port.

Inserting the security dongle will next open the Licences dialog box. Select Installation ID string with mouse and Right click and Copy the it into an email or notepad file. Then email it to Advanced Robotic Technology. They will email a licence file for you.

Meanwhile you can use the ToolShop in fully functional Trial mode. Please follow the steps below for activating ToolShop without a security dongle.

After you have received the license file, start the ToolShop with the security dongle plugged in and then select the licence file by using the "Select PC license" button in the Licences dialog box. After the license file has loaded click the OK button on the Licences dialog box. ToolShop will now start fully activated.

Hint: If the PC from which you are running ToolShop does not have email, copy and paste the Installation ID string into notepad. Then save the string as a text file into your USB key and email via another computer

Activating without a security dongle

Click the "Begin Trial" button. This will initiate the activation of ToolShop in Trial mode. The Trial mode requires that the PC has an Internet connection. Although the number of exports to the machine are limited to 30, there are no other limitations. You can request more exports for the Trial mode by contacting ART help line on 61 7 3393 6555 (ask for software support).

If you are running ToolShop in Trial mode and wish to start using the security dongle, simply plug in the security dongle to a USB port and then start the ToolShop. Then follow the instructions shown to request a license file. Meanwhile you can keep on using the ToolShop in Trial mode until you receive the license file via email.



Keyboard Shortcuts

This is a comprehensive list of available keyboard shortcuts in ToolShop program.

File Operations

Ctrl+O

Open

Ctrl+S

Save As

Ctrl+I

Import

Ctrl+E

Export

Ctrl+Z

Undo

Note! Undo will also work in Manual Nesting mode.

Standard Views

F2

Top View

F3

Left

F4

Front

F5

Bottom

F6

Back

F7

Right

F8

Diagonal SE

F9

Diagonal SW

F10

Diagonal NE

F11

Diagonal NW

Keyboard Options for Selection

Most keyboard options for selection are identical across ToolShop. However, there are some variations depending on the active mode.

Modes:

- **Select Mode**
All below keyboard options are accessible.
- **Layout | Manual Nesting Mode**
All below keyboard options are accessible.
- **Drawing | Node Edit | Select**
Only these keyboard options are available:
 - Shift+Mouse click**
Add or Remove node to/from selection.
 - Shift+Mouse marquee select**
Add or Remove a collection of nodes to/from selection.

List of keyboard options for selection:

Shift+Mouse click

Add or Remove shape to/from selection. .

Shift+Mouse marquee select

Add or Remove a collection of shapes to/from selection.

Ctrl+M

Select by Layer.

Ctrl+A

Select All.

Keyboard Options for Editing

Most keyboard options for editing are identical across ToolShop. However, there are some variations depending on the active mode.

Modes:

- **Select Mode**
All below keyboard options are accessible.
- **Layout | Manual Nesting Mode**
All below keyboard options are accessible, except “Ctrl+Mouse rotate part”. Also, in Manual Nesting mode editing will obey the Manual Nesting Gap and Edge Margin settings. So, movements and placements are restricted by the sheet boundaries and other shapes.
- **Drawing Mode**
Only these keyboard options are available in Drawing Mode:

Node Edit:

Arrow keys

Move selected nodes by Nudge Distance.

Shift+Arrow keys

Move selected nodes by 10x Nudge Distance.

Node Edit | To Arc:

Shift+Mouse click

After using “Node Edit | Select” to select the shape you wish to edit, click on the “Node Edit | To Arc”. Then you can change multiple line segments to arcs by holding down the *Shift* key while clicking on a line segment.

Node Edit | To Line:

Shift+Mouse click

After using “Node Edit | Select” to select the shape you wish to edit, click on the “Node Edit | To Line”. Then you can change multiple arcs to lines by holding down the *Shift* key while clicking an arc segment.

List of keyboard options for editing:

Arrow keys

Move selection by Nudge Distance. (Auto-repeat when keeping button down)

Shift+Arrow keys

Move selection by 10x Nudge Distance. (Auto-repeat when keeping button down)

Alt+Left Arrow key

Rotate selection counter-clockwise by Nudge Rotation value. (Auto-repeat when keeping button down)

Alt+Right Arrow key

Rotate selection clockwise by Nudge Rotation value. (Auto-repeat when keeping button down)

Ctrl+Mouse drag part

Leaves original and makes a duplicate at new location.

Ctrl+Mouse rotate part

Leaves original and makes a duplicate at new rotation.

Note! *This is not available in Manual Nesting mode.*

Angular Constraint Movement

You can adjust the polar angle in "Options | Preferences | Grid & Snaps | Polar Angle Settings".

Shift+Mouse drag part

Causes angular constraint movement.

Shift+Mouse rotate part

Causes angular constraint rotation.

Ctrl+Shift+Mouse drag part

Leaves original and makes a duplicate at an angular constraint location.

Ctrl+Shift+Mouse rotate part

Leaves original and makes a duplicate at an angular constraint rotation.

Ctrl+D (Duplicate)

Duplicates selection beside itself. The copy is placed on the right hand side and then made as the active selection. You can adjust the "Duplicate Gap" in "Options | Preferences | General Settings | Editing".

Del (Del)

Delete all selected objects on the screen.

Ctrl+L (Combine)

Combine will separate the parts into external and internal shapes. For the Combine to process the holes, the holes must be on the same layer as the outside shape. If the holes are required to be on a different layer, you may use the "Invert" function on the hole shapes to reverse the direction of the cut.

Note! *This was called "Create Hierarchy" in previous versions.*

Commented [JR33]: added

Ctrl+K (Break Apart)

Breaks apart combined shapes and returns the shapes to individual paths.

Note! *This was called "Remove Hierarchy" in previous versions.*

Ctrl+G (Group)

Groups selected objects into one entity.

Ctrl+Y (Smart Group)

Smart Group is a quick way of linking all objects contained within the perimeter of a parent object so it can be nested as a single part.

Commented [JR34]: added

Ctrl+U (Ungroup)

Ungroup the selected groups into individual entities.

ATP Module

The optional Automatic Toolpathing Module allows users to import multiple DXF files simultaneously. User can also specify materials and quantities for each file. In addition, user can choose from a list of options how the selected files will be processed. All the additional ATP module features are described in this section.

The ATP module is also designed to read a number of commonly used list-file formats such as Comma-Separated Values (CSV) in a .csv file, XLS, XLSX, and also DXT files generated by such programs as CabMaster. When importing list files the material and quantity property fields will be automatically filled in.

The ATP module uses the following fields from the list files:

- **FILENAME** – *The file to import*
- **QUANTITY** – *How many of each DXF file to import*
- **MATERIAL** – *The material to use*
- **THICKNESS** – *ToolShop recognises the thickness of material to use*
- **ROTATIONANGLE** – *A value set in degrees tells ToolShop to only rotate within a set angle. If you set this value to 0 then ToolShop will never rotate shapes in this file. Perfect for use with wood. For example, using value 180 is great if you wish to keep shapes horizontal. This means that the shapes in this file can have 2 rotation options. Using value 1 means that the shapes can have 360 rotation options. If the ROTATIONANGLE field has a value in the list file then ToolShop uses this value rather than the "Default Rotation Increment" value from the Nesting Template. If the ROTATIONANGLE field is left empty then ToolShop uses the "Default Rotation Increment" value from the Nesting template instead.*

After the list file is imported, these and all other fields will appear as shapes' custom properties within ToolShop. You can access the custom properties of a shape by pressing the right mouse button down on a shape.

To see all the generic fields available please see the section "An Introduction to Reporting and Labeling in ART ToolShop".

The ATP module can also read the material label from a DXF file. The material information is used to pre-fill the ATP dialog box. Please make sure that the **Preferences | Export Settings | Add Material Label (DXF Only)** setting is selected. This adds material information text field into DXF files exported from the ToolShop.

Commented [JR35]: added

Here is the list of ATP module additions.

File | Open (Ctrl+O)

File | Import (Ctrl+I)

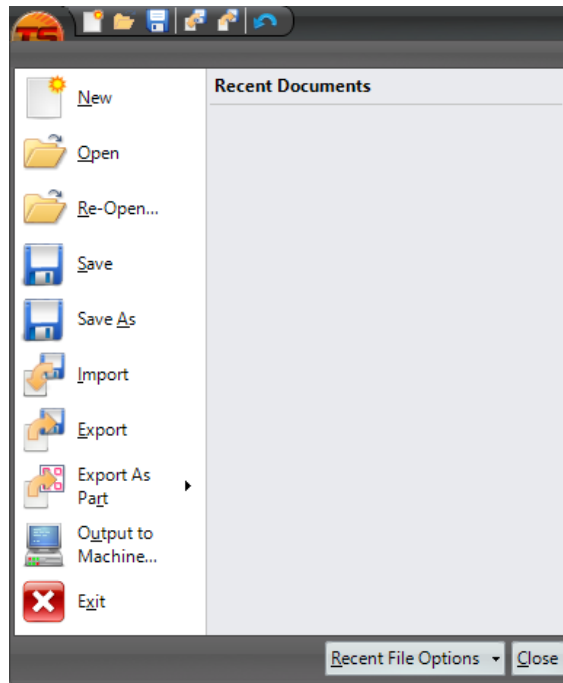


Figure: The File menu.

These functions open the standard file Open dialog box. To select multiple files hold down the Control key while you are clicking on the DXF files with mouse. Clicking the same file again will remove it from the file selection. After you have selected the files click on the Open button.

Note: The multiple file selection is only available for DXF, TSPART and list files.

Commented [JR36]: Pic updated

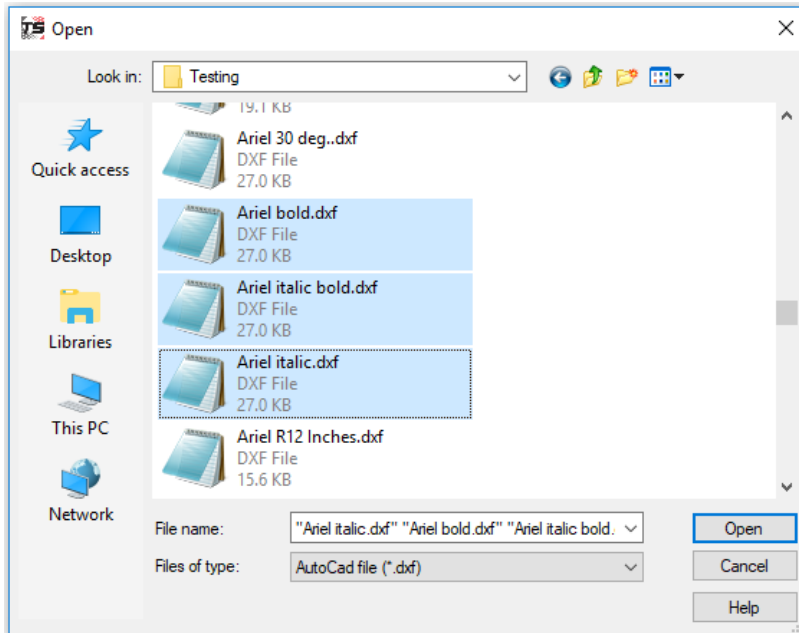
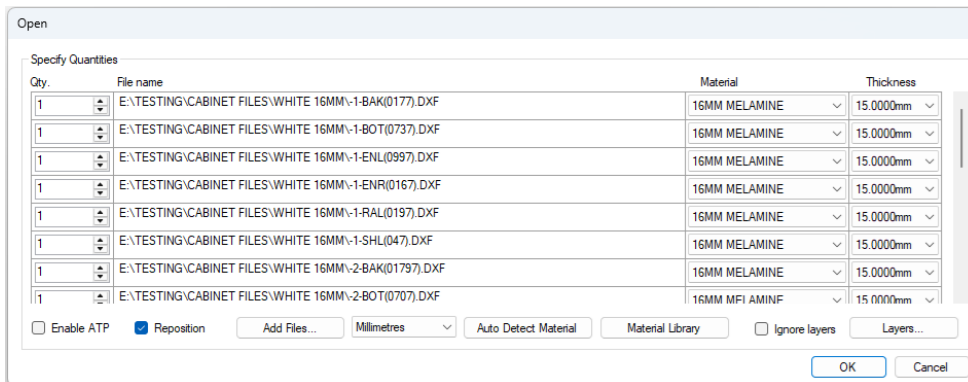


Figure: Users with an additional ATP module can open and import multiple DXF files simultaneously.

The Open button opens the Import dialog box where you can specify the quantities, materials, thicknesses and import options for the selected files.

File | Re-Open

The Re-Open function is similar to File | Open. However, the ATP dialog box is pre-filled with files from the previous ATP session.



Commented [JR37]: Pic updated

Figure: Specifying import options for the selected files

The Import dialog box options are as follows:

- **Enable ATP.** Use this option if you wish to further process the files. Enabling this option will show a list of available ATP options. Default value is OFF.
- **Reposition.** Choosing this option will stack the drawings on a vertical pile. The first imported drawing will be placed to X, Y coordinates 0,0. The next drawing will be placed on top of the pile etc. The copies of each individual file will be placed on the right hand side of the original along the same horizontal line. Default value is ON.
- **Add Files.** This opens the Open File dialog box again allowing you to add DXF or list files from other directories to your file selection.
- **Import Drawing Unit.** Select the drawing unit that was used in producing the DXF files. If the files were imported from a list file then ToolShop uses the material information from the list file. Select another drawing unit if the thicknesses appear incorrect. Then click Cancel and then open the list file again.
- **Ignore layers.** If this setting is ON then all shapes from DXF file(s) will be imported on layer 0. Default value is OFF which means that ToolShop will use the layers from the DXF file.
- **Layers...** button will open the Layer Manager dialog box. You can use this dialog box to choose which layers will be imported. The "Layers to be imported" tab page lists all the layers from the selected files. If **Enable ATP** option is OFF then only Visible, Locked and Import options are used.
 - **Layer Name.** Name of the layer.
 - **Material.** Used material.
 - **Thickness.** Material thickness.
 - **Toolpathing Template.** Applies predefined toolpath strategy to layer.
 - **Layer Visible.** Choose whether the layer will be visible after import.
 - **Layer Locked.** Specify whether layer will be locked.
 - **Layer Import.** Select the "Import" tick box for all layers you want to import.
 - **Layer Combine.** Perform Combine on the imported layer.
 - **Layer Invert Shapes.** Forces all shapes on layer to become internal holes.

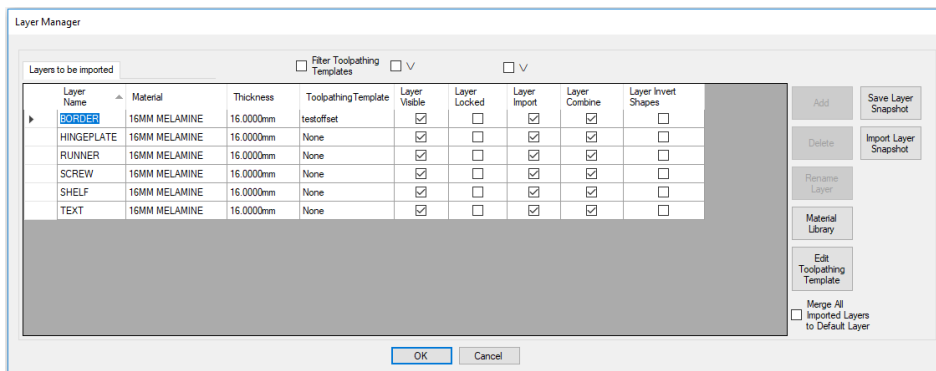


Figure: Selecting layer specific options via Layer Manager

Note: All imported layers and their settings will be automatically saved to ToolShop's internal layer library. Each time the Layer Manager is opened ToolShop checks from the layer library whether any of the layers have been imported in the past. For each layer that is found from the layer library ToolShop retrieves the latest stored values.

ATP Options (Enable ATP is ON)

- **Toolpath.** This will generate the toolpath for each imported layer by using the selected Toolpathing template for that layer.
- **Smart Group.** Groups together all shapes from individual file so that objects on different layers don't get separated during the nesting process.
- **Nest.** When shapes from all files have been imported then it will nest them.
- **Nesting Templates.** Allows you to edit selected template or create a new one.
- **Cut Order.** Selects whether cut ordering is done in rows or columns.
- **Output NC file.** Outputs toolpath as NC code.
- **Output DXF file.** Outputs each nested sheet as a DXF file.
- **Macros.** This will contain a list of available macros. They are sometimes used to add additional functionality to ATP. Macro files should be copied to the "Macros" sub-folder. If the Macro folder is empty then this option is not visible.

Commented [JR38]: Now called Smart Group

Qty.	File name	Material	Thickness
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-BAK(0177).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-BOT(0737).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-ENL(0997).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-ENR(0167).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-RAL(0197).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\1-SHL(047).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\2-BAK(01797).DXF	16MM MELAMINE	15.0000mm
1	E:\TESTING\CABINET FILES\WHITE 16MM\2-BOT(0707).DXF	16MM MFI AMINF	15.0000mm

☒ Enable ATP
 ☒ Reposition
 Add Files...
 Millimetres
 Auto Detect Material
 Material Library
☐ Ignore layers
 Layers...

Automatic Toolpathing
☒ Toolpath
☒ Smart Group
☒ Nest test Nesting Templates...
☒ Cut Order Columns Non-alternating, Left to Right, Bottom to Top
☒ Output NC file ☒ Output DXF file

Macros
☒ Merge sheets with gap Before output
☐ Merge sheets Before output
☐ Separate Layers Ver_3 Before output

OK Cancel

Figure: User is processing a group of files that were added from a DXT file. Preceding picture shows the layers from these same files.

Commented [JR39]: Pic updated, smartgrp

Importing part files

In addition to DXF files, you can also process ToolShop part files via the ATP module. The processing of part files (TSPART) differs from DXF files. The idea of a part is that it is all ready to cut.

When a part file is added to the ATP session, by default its material and thickness information is shown in the ATP dialog box. You can select a different material and thickness for the part by clicking with the mouse on the "Material" and "Thickness" drop down lists.

Commented [JR40]: added

When ATP processes part files the following options are bypassed:

Toolpath

Group

Macros (Before cleanup, After group/qty)

Commented [JR41]: added

The "Layers..." button allows you to choose actions that are executed for layers within each drawing file. The list of layers is populated from all the drawing files selected for the ATP session. This includes any layers from the part files as well. Layer actions will be bypassed for the part files but will be executed for the drawing files.

The "Cut Order" option is applied for each nested sheet. It takes into consideration each part's internal cut order. This guarantees that each part is processed first before proceeding to next part. Also, holes inside parts are cut before any parent closed shapes.

Commented [JR42]: added

Starting ATP session

When you click the OK button ToolShop will first perform a number of checks to determine if ATP can proceed. If the Toolpath option is ON then it will check that Toolpathing template has been defined for all imported layers. However, if you do not wish an imported layer to be toolpathed then simply select Template name "None" for that layer. ToolShop will show a warning "Toolpathing templates for the following layers are not assigned...Do you wish to continue?" Select "Yes" to continue. Clicking "No" will re-open the Layer Manager allowing you to alter the Template Names for the imported layers.

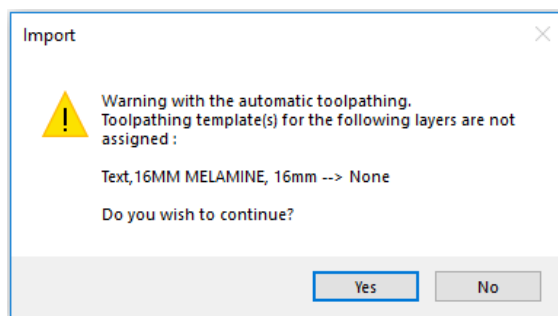


Figure: Layer "Text" has been assigned Template Name "None". It means that it will not be toolpathed. Select "Yes" to continue with ATP.

If the Toolpath option is ON then ToolShop will also check that the selected toolpathing templates are available and that the tools can be found from the Tool Changer settings.

Select the Nest option and nesting template if you wish ToolShop to perform nesting after Toolpathing. It is recommended that the Smart Group option is selected when nesting. This will make sure that all shapes in the individual file are grouped together before nesting. For example, an individual dxf might contain several shapes but put together they make up a single part.

Commented [JR43]: Was Group before

After performing the initial checks ToolShop will proceed with the ATP. An overall progress indicator is shown to the user. If you wish to cancel the ATP process you may click on Cancel button at any time.

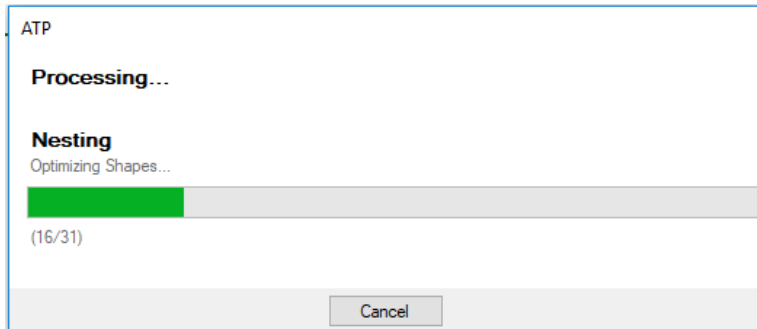


Figure: ATP process progress indicator.

If the Output NC file is selected then ToolShop will prompt user to input the NC filename at the end of the ATP process. You may abort the NC output by clicking the Cancel button.

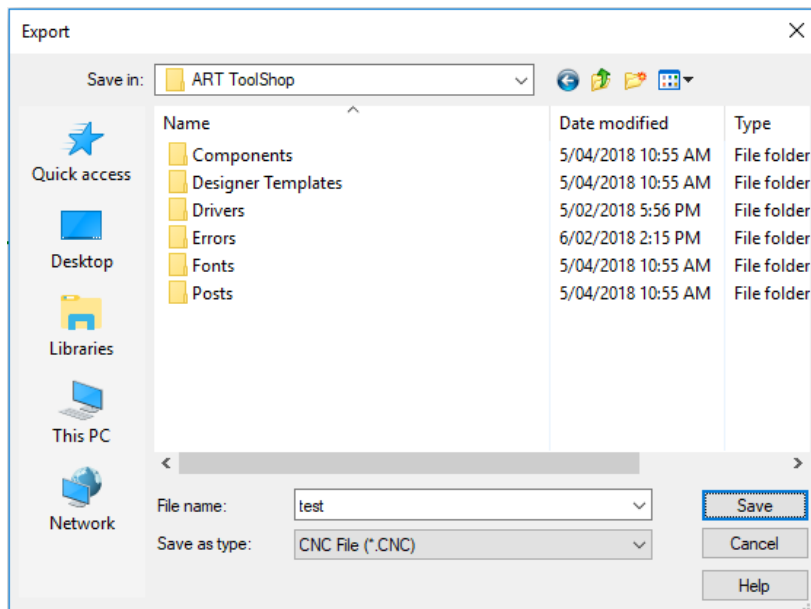


Figure: Output NC option will ask user to input filename at the end of ATP process.

To continue with the NC output to the selected filename please click the Save button. This will

open another Export dialog box that allows you to define which sheets are outputted. In the case of ATP it will automatically default to exporting "All Sheets". You may also override the cut order and print nest report and labels at this point. See the section "Output to Machine" in this document for more details.

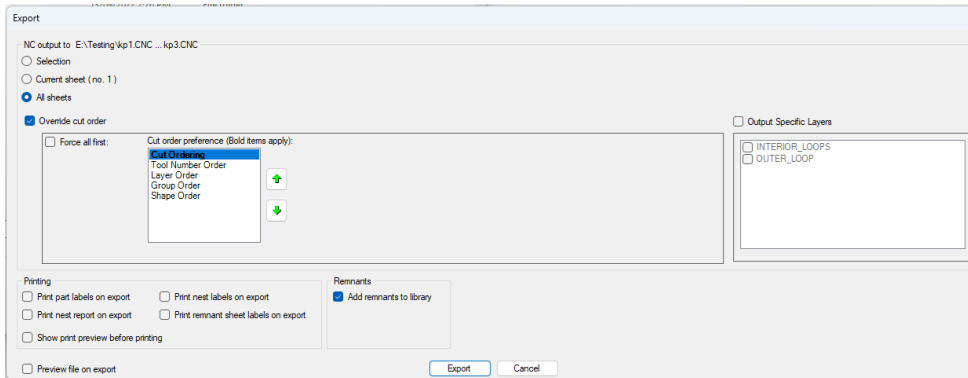


Figure: Output to NC options.

Note: When sheets are exported to NC files they are automatically deducted from the Material Library stock.

When ATP process is finished it will show the last processed sheet. If some of the shapes failed to nest due to material size or other reason, then user is shown the first sheet. In addition, the failed shapes are placed in a row outside the first sheet.

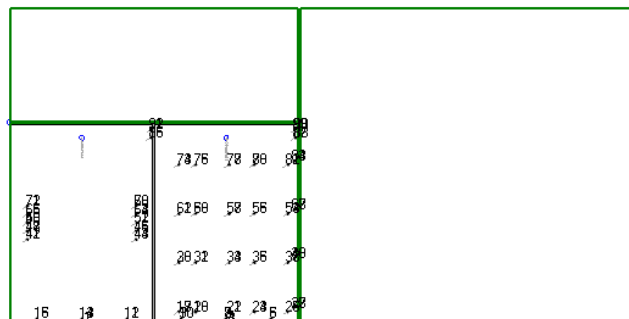
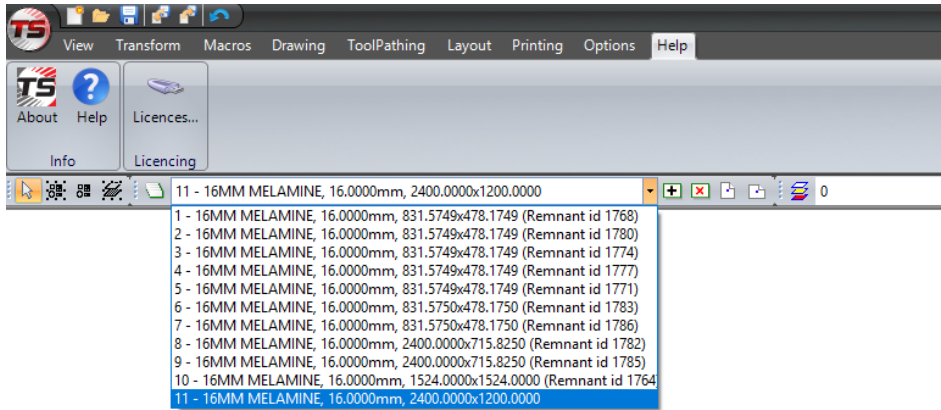


Figure: ATP process has used 11 sheets of 16mm melamine. The Nesting template that was used had the option "Use Remnants first" ON.

Move to Layer

If ATP module is enabled then "Layers" drop-down list box lists all the layer names from the layer library. If ATP module is not enabled then only layer names from current job are listed. See the section "Moving selected shapes to another layer" for details how to use this function.

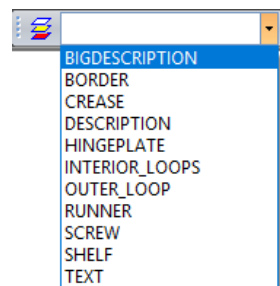


Figure: Layers drop-down list box with ATP licence.

Printing

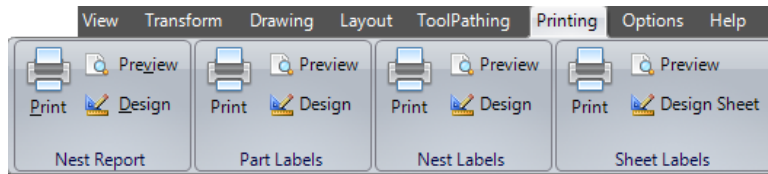


Figure: The Printing menu with ATP licence.

ToolShop uses a similar designer for all print documents. The difference is that each of the designers may have slight variations in the items available, as well as variations in the way they're output. To learn the basics of using the designer, read the section "Using the Designer" in "An Introduction to Reporting and Labeling in ART ToolShop".

When you open any of the print document designers they will start with current designer template. If no current designer template is defined or the default designer template is missing for the print document then user is prompted to create a new template.



Nest Report

See the section "Nest Report" under "Printing menu" for more details on the nest reports.



Part Labels | Design

The Part Labels Designer allows user to design a part label template. Upon entering the designer user is prompted to "Select Printer", if required.

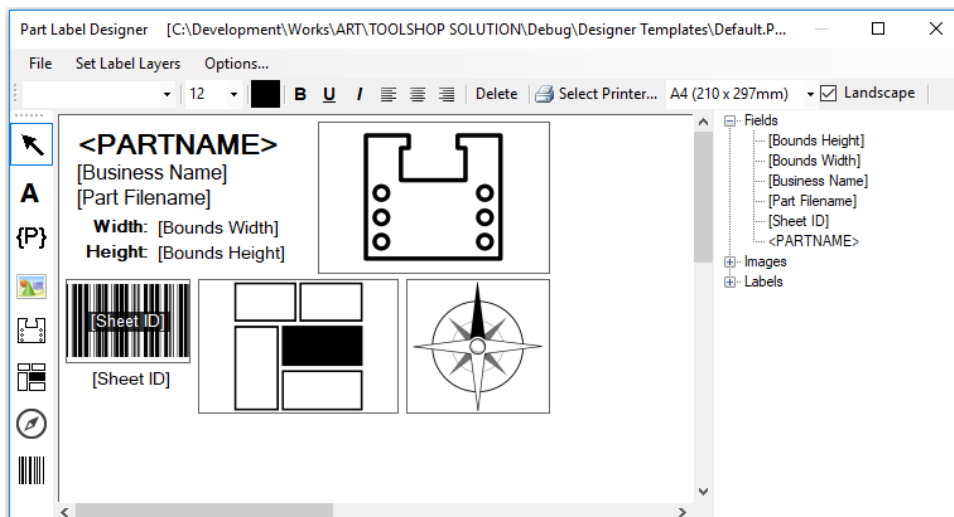


Figure: User has opened part label template "Designer Templates \ Default.PartLabelLayout.TSPartLbl".

Set Label Layers

In order to set which layers are marked Part Label layers click the menu at the top saying “*Set Label Layers*”. You will then be presented with a list of layers with tick boxes. Ticking a layer name will assign that as a layer to find parts on.

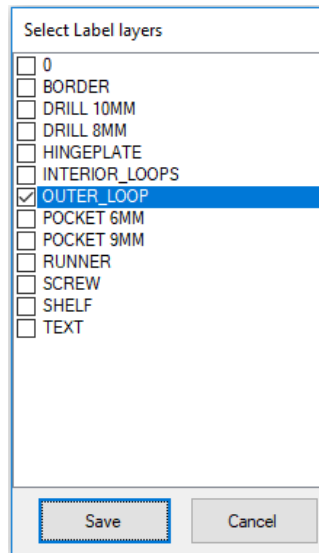


Figure: Part labels are printed for parts found on selected layers.

Symbols

Below is a description of the special symbols available on a Part Label.



Shape Preview

The Shape Preview gives operators the ability to identify at a glance, the correct part for the Part Label. A scaled preview of the shape is printed onto the label to fit the box that was placed on the label designer.



Part Compass

The Part Compass on a label applied horizontally onto the nest will point in the direction of the parts original north. For example, if a part after being imported was rotated 45° degrees clockwise, the compass would point north-east.



Nest Indicator

As parts can often be very similar the nest indicator gives you a preview of the entire nest, with the labels part filled with black. This makes identifying an area of similar parts a breeze.

{P} Fields

Below are the fields specific for Part Labels. To see all the generic fields available see the section "Using the Designer" in "An Introduction to Reporting and Labeling in ART ToolShop".

[Part Filename] – The filename of the DXF file.

[Bounds Width] – The maximum width of the rectangle bounds.

[Bounds Height] – The maximum height of the rectangle bounds.

[Bounds Area] – The area used by the rectangle bounds.

[Part Sheet #] – The sheet number the target part is on. (Usually 1, 2, 3, 4...)

{P} Custom Property Fields

In addition to standard fields, the Part Labels have the ability to print the Custom Properties provided by a shape. This allows you to have a lot more information on each Part Label. Each of the Custom Properties within a shape will automatically appear in the "Select Fields..." dialog box in the designer. These values will have arrow brackets (<>) surrounding them.

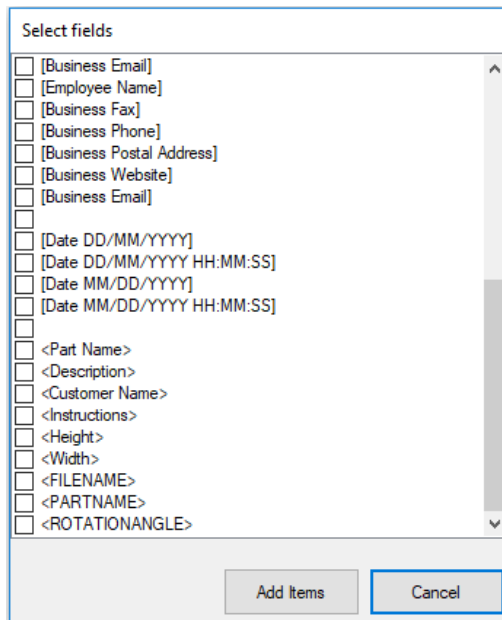


Figure: The custom property fields (<property>) appear at the end of the "Select fields" list.

Note: The values for the custom properties, e.g. <Part Name> and so on need to be provided by the shape. ToolShop's ATP module is able to use list files (*.LST) by adding any text fields

that were attached to parts as a Custom Property. Alternatively you can add or edit custom properties by right clicking on selected shapes in ToolShop main window and selecting "Custom Properties..." from the popup menu.

For example, if you wish to add the custom property field "<Part Name>" into your template in Part Labels Designer, you will also need to have the property "Part Name" with a value in the custom properties of the printed shapes. If a shape is missing a custom property and/or its value then it is not shown on the "Part Labels Preview" window or paper print out.



Part Labels | Preview

This function enables user to preview the part label before printing. First user is prompted to select the printing range. The available options are "All Sheets", "Current Sheet" and "Selected shapes only". Select the printing range and click OK button. ToolShop now opens the "Part Labels Preview" window.

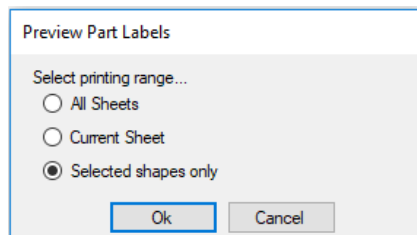


Figure: Select printing range for Part Labels Preview.

It is important that during the designing of the part label template user selects the part label layers. Part labels are printed for every 'group' of shapes in your ToolShop nest on these layers. If the part label layers are not defined then the "Part Labels Preview" window appears empty and an error message is shown.

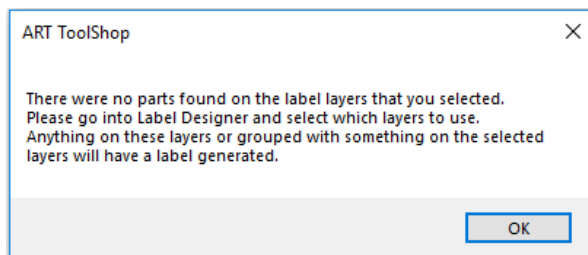


Figure: Part labels cannot be previewed or printed if part label layers were not selected during the designing of the part label template.

Click "Print" button on the "Part Labels Preview" window to send the document directly to the default printer. To exit the preview window click "Close" or the "X" on the top right corner.

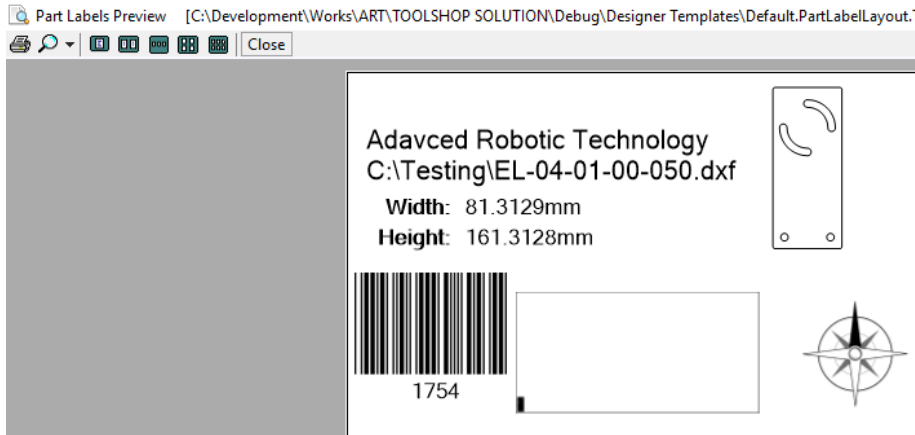


Figure: Preview of the current part label “Designer Templates \ Default.PartLabelLayout.TSPartLbl”.



Part Labels | Print

This function opens the “Print” dialog box which allows user to print the part labels. Select the appropriate printer and settings and click the “Print” button.



Nest Labels | Design

This function opens the Nest Label Designer window which allows user to design a nest label template. Upon entering the designer user is prompted to Select Printer, if required.

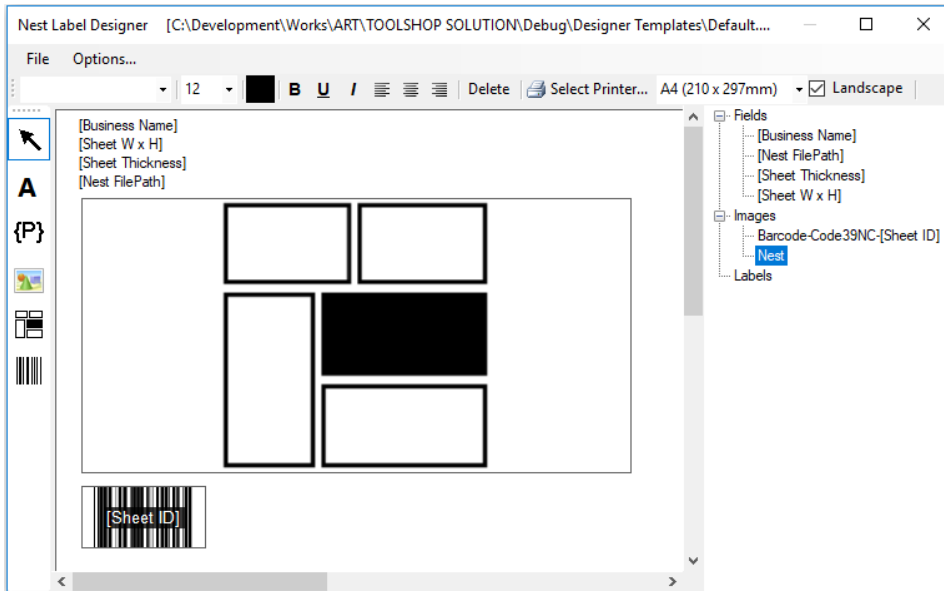


Figure: User has opened nest label template "Designer Templates \ Default.NestLabelLayout.TSNestLbl".

Symbols

Nest Labels have the same symbols as those available on a Nest Report.

{P} Fields

Nest Labels have the same fields as those available on a Nest Report.



Nest Labels | Preview

This function enables user to preview the nest labels before printing. First user is prompted to select the printing range. The available options are "All Sheets", "Current Sheet" and "Selected shapes only". Select the printing range and click OK button. ToolShop now opens the "Nest Labels Preview" window.

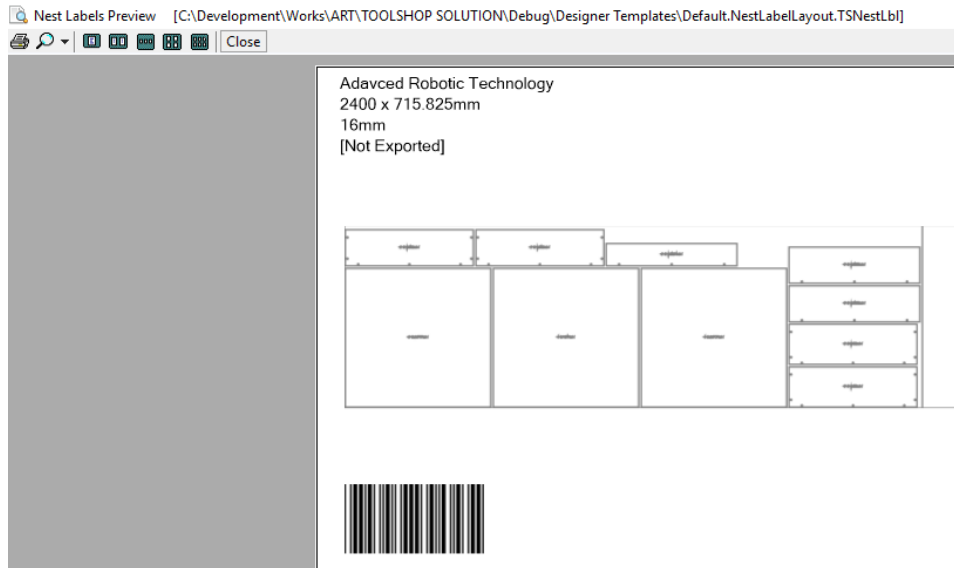


Figure: Preview of the current nest label "Designer Templates \ Default.NestLabelLayout.TSNestLbl".



Nest Labels | Print

This function opens the "Print" dialog box which allows user to print the nest labels. Select the appropriate printer and settings and click the "Print" button.



Sheet Labels | Design

The Sheet Labels Designer allows user to design a sheet label template.

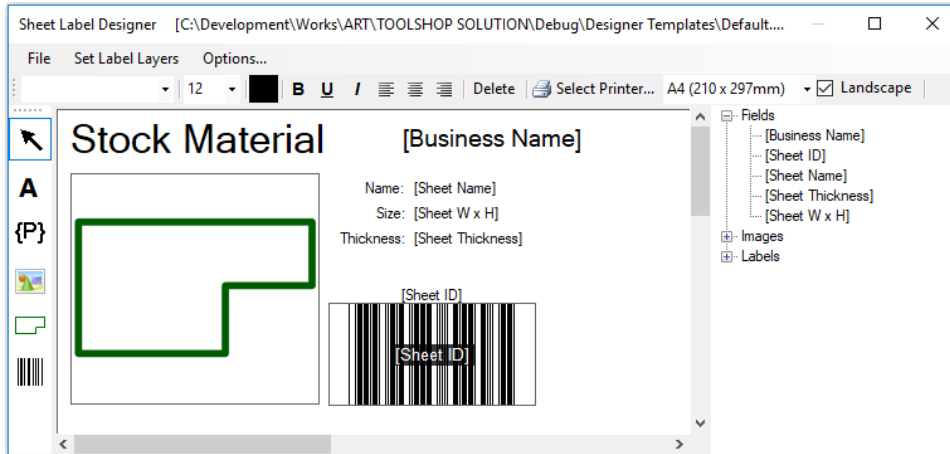


Figure: User has opened sheet label template "Designer Templates \ Default.SheetLabelLayout.TSShtLbl".

Symbols

Below is a description of the special symbols available on Sheet Labels.



Sheet Shape

The sheet symbol is an effective way of identifying sheets by its height/width ratio and also if it's an oddly shaped remnant. The Sheet Shape symbol will print an outline of the sheet.

{P} Fields

Below are the fields mostly used for Sheet Labels. To see all the generic fields available see the section "Using the Designer" in "An Introduction to Reporting and Labeling in ART ToolShop".

[Is Remnant?] – This prints either 'Yes' or 'No' depending on whether the target sheet is a remnant.



Sheet Labels | Preview

This function enables user to preview the sheet labels for the current job before printing.

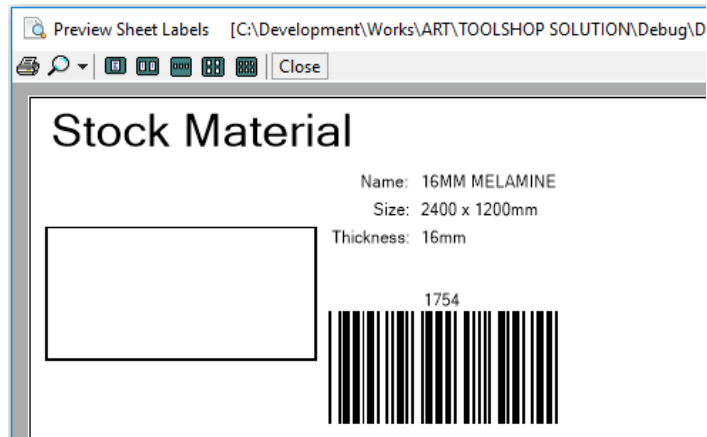


Figure: Preview of the current sheet label “Designer Templates \ Default.SheetLabelLayout.TSShtLbl”



Sheet Labels | Print

This function opens the “Print” dialog box which allows user to print the sheet labels for the current job. Select the appropriate printer and settings and click the “Print” button.



Options | Preferences

If ATP module is enabled then the “Import Settings” tab page on the Preferences dialog also contains some ATP specific global settings. These global settings determine which options are selected as default on the Import dialog box.

Part of the ATP module is the ability to import CSV, DXT, XLS and XSLX list files. These contain a list of DXF files and properties. If there is no ‘Filename’ specified in the list file, ATP will automatically use the specified DXF instead.

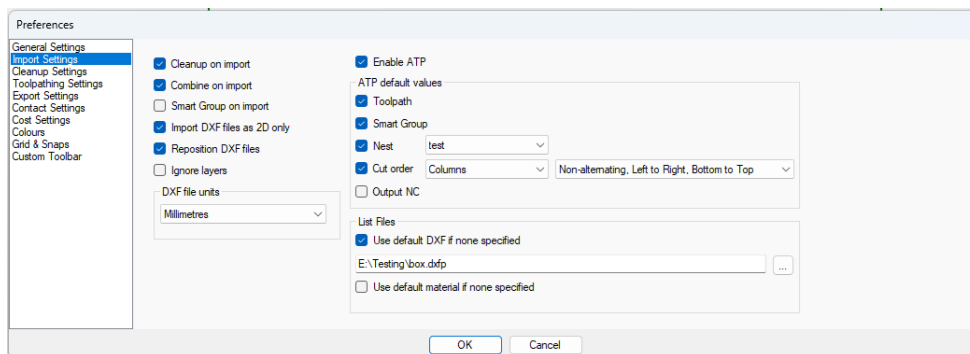


Figure: Import Settings specific to the ATP module.

Commented [JR44]: Pic updated, smartgrp, cut order style dropdown



Options | Material Library

If ATP module is enabled then the Material Library allows you to print stock labels for materials. You can also define Material Templates which are default toolpathing templates for specific materials and thicknesses.

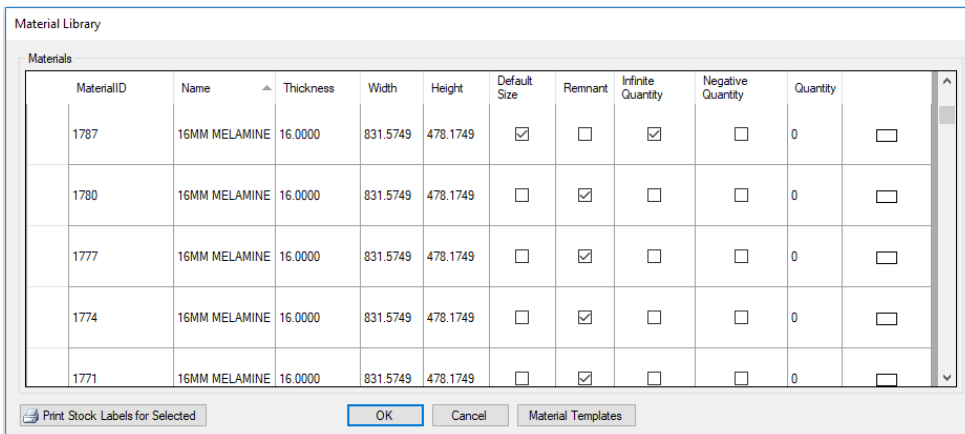


Figure: Material Library window with ATP licence.



Print Stock Labels for Selected

If you are just setting up ToolShop or if a new material is added to the material library, Sheet Labels can be printed for these new materials. Within the Material Library, simply select a single row, or select multiple rows by holding CTRL while selecting. Then click the

Print Stock Labels for Selected button to print these labels.

Also, if a remnant is added to the stock, a single Sheet Label could also be printed for this. However, usually this would be done at the time of exporting.

Material Templates

By clicking on the "Material Templates" button allows user to define default toolpathing templates for specific materials and thicknesses. Select appropriate material and thickness from the drop down lists and then choose the template. The availability of templates depends on the selected machine configuration. At the moment Hidef Plasma configurations come with pre-defined toolpathing templates. However, you can add your own templates via any of the Toolpathing dialog boxes.

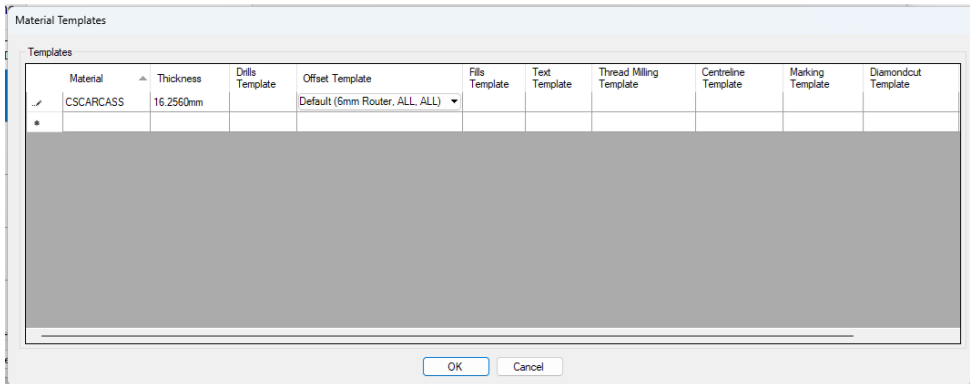


Figure: The Material Templates dialog box allows user to define [Default] templates for specific materials and thicknesses.

Commented [JR45]: Pic updated, marking template column added

After defining the material templates you can use them during the ATP process. By clicking the “Layers” button on the ATP dialog box you can select toolpathing templates that are used for each imported layer. In order to use the material templates, you will need to select one of the “[Default xxx]” items. For example, if you wish to use the default Offset template for 10.000mm STAINLESS, then select from the list “[Default Offset]”. The ATP process will now use the material template that was assigned as the Offset Template for the 10.000mm STAINLESS to toolpath that layer. The ATP process will automatically import all imported layer names along with material and template information into the Layer Library. You can change the toolpathing templates for already imported layers via Layer Manager.

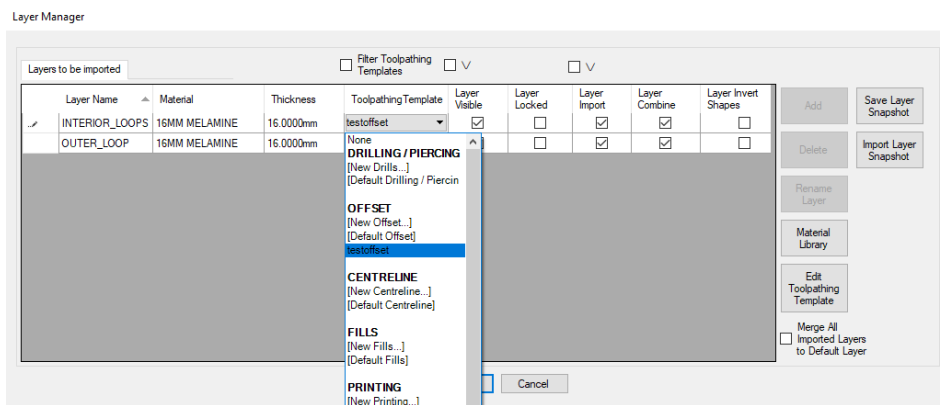


Figure: Using the “Default Offset” material template for 10.000mm STAINLESS during ATP process.

Defining material templates for the most frequently used materials is of real benefit when using ATP. If no specific toolpathing template has been defined for any of the imported layers then ToolShop will suggest appropriate template based on the selected material and thickness from the Material Templates definitions.



Options | Layer Manager

If ATP module is enabled, then the Layer Manager contains additional settings for each layer. For example, you can assign toolpathing templates for different layer, material and thickness combinations. The user is also able to browse the current layer library and add or remove layers via the "All Layers" tab page. The "All Layers" tab page contains all layers with their default values that have been imported to ToolShop. You can change these default settings to your liking.

If the "Filter Toolpathing Templates" option is selected then the "Toolpathing Templates" dropdown list will show only templates that are valid for the selected layer, material or thickness. When saving toolpathing templates user can specify the material that it is linked to. Alternatively, template can be saved with ALL materials enabled or with a specific thickness allowed.

Above the "Visible" and "Import" columns is a tick box that allows you to select or deselect the "Visible" and "Import" option for all existing layers.


Both Material and Thickness drop-down lists have selection called "ALL". For example, if we have layer BORDER with Material = "16MM MELAMINE" and Thickness = "ALL", this means that toolpathing template called "5mm below surface" is applicable for any thickness of that material. However, you will need to make sure that template's Depth Relative Location is appropriate e.g. "Below Surface".

You may edit the selected template by clicking on the "Edit Toolpathing Template" button. If you wish to add new templates, select from the Toolpathing Template drop-down list item [New Offset...] or [New Centreline...] etc. To make use of the Material Templates select item [Default Offset...] etc.

You can also access the Material Library from this dialog box by clicking on the Material Library button.

Layer Name	Material	Thickness	Toolpathing Template	Layer Visible	Layer Locked	Layer Import	Layer Combine	Layer Invert Shapes
BIGDESCRIPTION	PLYWOOD	19.0500mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
BORDER	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
	PLYWOOD	19.0500mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
CREASE	PLYWOOD	19.0500mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
DESCRIPTION	PLYWOOD	19.0500mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
HINGEPLATE	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
RUNNER	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
SCREW	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
SHELF	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
TEXT	16MM MELAMINE	16.0000mm	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Figure: Layer Manager dialog box when ATP module is enabled.



The "Merge All Imported Layers to Default Layer" option is available when selecting which drawing layers are imported via ATP Import dialog box. This option will allow user to overwrite drawing specific layer information. For example, if this option is enabled and you have selected in ATP Import | Layers dialog box to import two layers "inner shapes" and "outer shapes" from a specific drawing, then all shapes on those layers are imported onto default "0" layer. This option is practical when the user wants to treat all imported layers the same way and has no need for the layer information. This could be the case when dxf drawings are sourced from other engineering companies.

The "Save Layer Snapshot" command will allow you to save all layers and their settings in the current layer manager view into a layer snapshot file. This could be for backup reasons or when transferring layer definitions onto another PC. The saved layer settings can be re-imported later using "Import Layer Snapshot" command.



Producing Quality Drawings

In order for ToolShop to be able to process your CAD drawings correctly, it is vital that they are of a good quality. Unfortunately, many cad drawings are produced solely for the purpose of printing. While these may be quite legible in printed form, it is not uncommon to find multiple lines on top of one another. Another problem often found in DXF files is corners that do not connect. These are often the result of the methods used in constructing the original drawing.

Duplicate lines...

For instance, you may draw a rectangle. Later, you may apply a fillet or rounding to the corners of the rectangle. If you were not careful it is possible in some programs to apply a fillet twice to one corner. This would result in two curves, exactly the same, one on top of the other.

Duplicate shapes...

It is not uncommon to find that an object has been duplicated on top of itself. This may be the result of an array, copy or mirror function that was performed during the course of the drawing. Typical results of this error include dual toolpaths on what appears to be a single object as well as objects that refuse to toolpath correctly. You would usually notice that the part is incorrectly identified as an outside shape when it is actually a hole. Another case is when a shape will not be applied as a parent object, in the case of a part with holes or other internal features.

Open corners...

As previously mentioned, another issue that can hinder the correct processing of a CAD file is open corners. This can occur if a line is drawn freehand without snaps enabled. Other causes include accidentally deleting a very small segment that was part of a curve or other complex contour. You must ensure that all lines are connected even when you zoom up close.

Unsupported entities...

The DXF file format has been in existence for a number of years and has undergone many changes during this time. As new drawing processes have been invented the DXF format has been modified to support this new object. Some entities that have been added include Polylines, Splines and Meshes to name but a few. While every effort has been made to provide support for the most widely used entities, many are not applicable or desirable. If you find that an entity (a line or curve etc.) disappears when it is transferred into ART ToolShop then you may have to use the EXPLODE method inside your CAD program on the object before you export it as a DXF drawing. This has the effect of reducing the entity to a more basic form of lines and arcs. You will find many issues of file compatibility are solved by using this option.

Automated Design Applications...

Over the years we have come across a few automated design applications that will generate DXF drawings for you. These applications, while extremely useful, often produce lower quality drawings. Some pitfalls can be reduced accuracy, open corners, chunky curves (made up of large flat segments) and other issues. If you experience such difficulties it may be necessary to open these first in another CAD program for manual cleanup or redrawing.

Scale...

It is very important that your drawings are exported in METRIC at 1:1 ratio. If you draw in another scale or unit such as feet/inches it is important that the software is set to scale the drawing on output to ensure that the final result is not scaled to some other dimension. If you have any difficulty with this you should consult the documentation that comes with your CAD software.

Some options to solve scaling issues with software that does not support metric output are as follows:

- Set the DXF output driver to output in metric
- Draw in a scale of 1:25.4
- Rescale the drawing before you output the DXF file

We hope this short guide will enable you to produce quality drawings that transfer flawlessly. If you have any continuing issues, please contact the ART help line by phone on 61 7 3393 6555 (ask for software support) or your local distributor for help.



Step by Step Guide to Cutting a Job.

Use the following guide as a general guide for processing a job in ART ToolShop:

Step 1: DXF File

The simplest method is to create a CAD drawing with all of the components that you wish to cut in the one drawing. This should then be exported as a DXF file from the CAD program. It is good practice to create a folder on your PC for the express purpose of saving these files.

Step 2: Open ART ToolShop

Double click on the ToolShop icon on the desktop or select ART ToolShop from the Start/All programs menu



Figure: Double-click on this icon to start ART ToolShop

Step 3: Select File | Open menu

Select File | Open menu or click on the Open icon on the toolbar



Figure: Click this icon on the File menu or the quick launch toolbar to open a DXF file.

Step 4: Select All

Click on the 'Select All' icon on the Transform menu to ensure that you are modifying all parts on the screen



Figure: Click this "Transform | Select All" icon to select all parts

Step 5: Cleanup selected

Click on the 'Cleanup selected' icon on the Transform menu to clean up the DXF drawing. This will join corners and convert segmented arcs into smooth curves.



Figure: Click this "Transform | Cleanup Selected" icon to clean up the drawing

Step 6: Combine

Click on the 'Combine' icon on the Transform menu to automatically sort out which shapes are holes and which shapes are outside contours. For the Combine to process the holes, the holes must be on the same layer as the outside shape. If the holes are required to be on a different layer, you may use the "Invert" function on the hole shapes to reverse the direction of the cut.

Commented [JR46]: Combine function change in behaviour.



Figure: Click on this “Transform | Combine” icon to sort inside and outside shapes

Step 7: Toolpathing

Click on the 'Offset Toolpath' icon on the Toolpathing menu to specify the tool parameters.



Figure: Click on this “Toolpathing | Offset” icon to generate an offset toolpath

You will need to fill in the following information for Plasma cutters:

Tool Number:

Choose between the tools available from the tool changer menu.

Cut Width:

Enter the width of cut for the main tool. If required, accurately measure a test cut in the material using the selected tool to find this value.

XY Feed rate:

Set the feed rate according to the desired cut speed as outlined in the Hypertherm manual for the selected material.

Pierce Height:

Set this according to the Plasma manual for this tool/material combination.

Lead-In length:

Set this to a distance that is greater than the pierce hole diameter so that the piercing process does not damage the parts.

Pre-Piercing:

Set to TRUE if you require the machine to do a pierce with a different tool to the one that is being used for the main cutting. You will need to set the following settings with this function.

Pierce Tool Number:

Select the tool to use for the pre-pierce.

Pierce Delay

Set the delay period for the tool from the plasma manual

Pierce Height

Set the pierce height for the tool from the plasma manual.

Step 8: Auto Nest

Click on the 'Auto Nest' icon on the Layout menu. This will fit the parts into the material as accurately as possible. Some manual manipulation of parts may be required if outcome is not as desired. You can use the Manual nesting function to accomplish this.



Figure: Click on this "Layout | Auto Nest" icon to nest the parts into the material

Step 9: Edit Start Points

Click on the 'Edit start points' icon on the Toolpathing menu. This will highlight all the start point with a diamond shape. You can then drag the diamond to any point on the shapes. This is vital to ensure that the machine does not pierce on top of another shape.

When you are satisfied that all the pierce points are away from other objects you can use the right mouse button to exit the function.



Figure: Click on this "Toolpathing | Edit Start Points" icon to edit start points

Step 10: Cut Order

Click on the 'Manual Cut Order' icon on the toolbar to adjust the cut order of the parts. This is important to save time in cutting (less traversing) and to keep the heat even in the plate. Simply drag a line around the drawing in the order that you wish the parts to cut. When you have finished, use the right mouse button to exit the function. The program will display the cut order numbering of each part. If you are not satisfied with the result, you can redo this process as many times as required to get the result you want.



Figure: Click on this "Layout | Cut Order | Manual" icon to select the cut order

Step 11: Export CNC

Click on the 'Export' icon on the File menu or quick launch toolbar to create the CNC file required by the machine. You should save this file into a folder on your PC and then copy it to the machine using either your network, or a USB memory stick. Follow the directions in the ART ProfileShop manual to further process the job on the machine.



Figure: Click on this "File | Export" icon to create the CNC file that is sent to the machine.





ToolShop User Manual

About this document

This document is designed to introduce you to the reporting and labelling features that are now available in ART ToolShop.

These features are usually used alongside the Automatic Tool Path (ATP) feature for the purpose of tracking materials and parts, while also providing useful information to aid in the estimation and quotation process.

Each section in this document is designed to explain how you can easily make use of each feature, as well as guiding you through the process of getting started.

In this document there are six sections relating to:

1. *Creating list files for automatic importing of components.*
2. *Using Nest Reports to assist machine operators or sales quoting.*
3. *Automatic job loading using Nest Labels.*
4. *Managing sheet stock through the use of Sheet Labels.*
5. *Intelligent tracking of individual parts using Part Labels.*
6. *Using the designer.*

As we go through each of these six sections, you will greatly expand your knowledge of these new ToolShop features.

Specific Terminology

Part – When mentioning a part, this document refers to the component that was exported from CAD/CAM software and then cut on CNC machinery.

Shape – This document refers to Shapes as the virtual part which is currently a series of lines on the screen. DXF files contain one or more components. Toolpaths are then applied to shapes before export.



Index

1 Using List Files for Automatic Importing of Shapes

- 1.1 How details from list files are used in labelling
- 1.2 File types and formats
- 1.3 Custom Properties

2 Using Nest Reports to Assist Machine Operators & Quoting

- 2.1 Designing a Nest Report
- 2.2 Symbols
- 2.3 Fields

3 Reduce Human Error using Nest Labels

- 3.1 Using Nest Labels for pre-allocating and picking material before cutting process
- 3.2 Bar-codes for automatic job loading at the CNC machine
- 3.3 Using nest preview thumbnail to verify correct CNC file

4 Sheet Labels

- 4.1 Track Stock with Sheet Labels
- 4.2 Printing sheet labels at time of receiving stock
- 4.3 Printing Sheet Labels at time of export
- 4.4 Symbols
- 4.5 Fields

5 Part Labels

- 5.1 Why use part labels?
- 5.2 Examples of using Part Labels Effectively
- 5.3 Symbols
- 5.4 Custom Property Fields
- 5.5 Fields

6 Using the Designer



ART ToolShop

6.1 Opening the designer

6.2 Page & Printer Setup

6.3 Generic Symbols

6.4 Generic Fields

1. Using List Files for Automatic Importing of Shapes

How Details from List Files Are Used In Labelling

Third party software, such as cabinet design software, exports a collection of DXF files containing part drawings. Along with these DXF files, a formatted list is saved that describes how many times each part is required to be cut.

ToolShop has the ability to import these formatted lists that contain filenames, quantities and text fields for each of the DXF files.

ToolShop's ATP module is able to use these list files by adding any text fields that were attached to parts as a Custom Property.

Custom properties are then used in [Part Labels](#) which are discussed later in this document.

File Types & Formats

The ATP module is already designed to read a number of commonly used list-file formats such as Comma-Separated Values (CSV) in a .txt file, XLS, XLSX and also DXT files generated by such programs as CabMaster.

Custom Properties

As mentioned earlier, ToolShop is able to import text fields attached to DXF's within the list file.

ToolShop has been made to recognise some specific fields such as:

- **FILENAME** – *The file to import*
- **QUANTITY** – *How many of each DXF file to import*
- **MATERIAL** – *The material to use*
- **THICKNESS** – *ToolShop recognises the thickness of material to use*
- **ROTATIONANGLE** – *A value set in degrees tells ToolShop to only rotate within a set angle. 0 rotation angle tells ToolShop to never rotate shapes in this file. Perfect for use with wood.*

Any other custom field is attached to each shape as a 'Custom Property'. These Custom Properties can then be identified via the Label Designer and be placed onto the designer showing the attached value when printed. For example; PartName, CustomerName, CustomerPhone placeholders could all be dropped onto the designer and then their corresponding value will be printed from each shape.

In order to edit Custom Properties of a shape, you will need to go to the main display window and select the shape(s) in which the properties should be edited. Once the shape(s) are selected, right click and select "Custom Properties..." from the popup menu.

A dialog box will then appear with some default suggestion properties as well as anything that was imported from a list file or previously added by you. You may select multiple shapes at a time and edit their properties simultaneously.

To add a new custom property, simply start typing in the bottom row.

Edit Part Properties

Property Name	Property Value	Change
ALLOWROTATE	1	<input type="checkbox"/>
FILENAME	C:\Testing\Cabinet files\White 16mm\1-EnR(01...	<input type="checkbox"/>
LENGTH	750.0000	<input type="checkbox"/>
MATERIAL	16mm Melamine	<input type="checkbox"/>
MIRRORED	0	<input type="checkbox"/>
PARTNAME	dave Floor Standard Cabinet #1 "Floor Doorx1" [...	<input type="checkbox"/>
QUANTITY	1	<input type="checkbox"/>
ROTATIONANGLE	0.000000	<input type="checkbox"/>
THICKNESS	16	<input type="checkbox"/>
UNITNUMBER	1	<input type="checkbox"/>
WIDTH	542.0000	<input type="checkbox"/>
**		<input checked="" type="checkbox"/>

Change Properties Cancel

Rows in grey have not been edited and won't change the value in the selected shape(s). This is because multiple shapes may have a different value for things such as shape name, but you might only want to change their customer names. Therefore, only rows that you edit will get changed. Also, unchecking the "Change" tick box will cancel the changes for that row.

2. Using Nest Reports to Assist Machine Operators & Quoting

Nest Reports are a handy feature within ToolShop that allows you to print graphical job reports with detailed information available from a selection of fields.

The purpose for Nest Reports is most common for two areas of a business.

First off, a big part of any cutting for hire business is that of needing to generate accurate quotes. The information available on a Nest Report is highly detailed with enough information for you to easily generate quotes that are highly accurate in both the estimated process time and estimated cost.

Secondly, through the use of Nest Reports, operators are easily able to identify that they have loaded the correct file, check that they have the correct tooling, as well as exactly what material is needed. These can especially help when an operator may have spare time between loading jobs as no computer is needed to get information on the nested job.

Designing a Nest Report

Firstly, open the 'Nest Report | Design' under the Printing menu. Once clicked the Nest Report Designer will open.

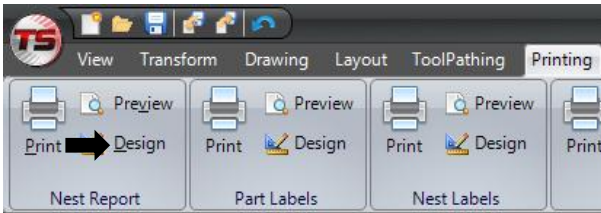


Figure: Click on the “Nest | Report” icon to open the Nest Report Designer.

As we are usually designing for a larger sheet of paper, it is easier if we make our window go full screen.

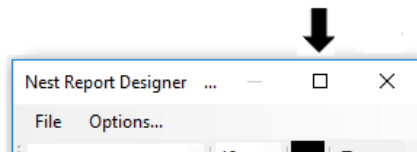


Figure: Switching to the full screen mode can be helpful when designing nest reports.

To learn the basics of using the designer, read [6. Using the Designer](#).

Symbols

Below is a description of the special symbols available on a Nest Report.



Nest

When a nest symbol is placed onto the designer, it will show a representation of the nested shapes once the Nest Report is printed.



Figure: A typical representation of nested shapes. This bitmap is shown in place of the nest symbol.



Sheet Shape

The sheet symbol is an effective way of identifying sheets that are remnants. It will place an outline of the sheet onto the nest report.

Fields

The Nest Report has a number of fields available that are specific to the Nest Report.

[Tools Used] – Provides a simple multiline list, containing one tool number and name per line.

6: 6mm Drill

7: 9mm Drill

[Tools Used Summary] – Generates a table at the upper left corner of this field. The table contains a list of all

the tools used, followed by values such as: Tool name, number of starts, estimated time cutting, cut distance, time spent traveling between shapes and total distance of travel between shapes.

Tool	Starts	Cut Time	Cut Distance	Travel Time	Travel Distance
6mm Router	67	9.13min	1438.2131"	1.75min	1651.0046"

[Total Cut Length] – The total distance the tool will be cutting. This field will use the same measurement (mm, inches) unit as the ToolShop interface.

[Total Cut Time] – The estimated time the tool will spend cutting.

[Sheet Area Used] – Outputs a number representing the total area covered by parts on this nest. This is followed by the percentage of the actual area used.

[Sheet Number] – Prints the index of this sheet in relation to the job file. Eg, Sheet 1, 2 or 3.

[Is Remnant?] – This prints either 'Yes' or 'No' depending on whether the target sheet is a remnant.

[Nest Filename] – This is the filename of the ToolShop nest file. Eg. "Example_Nest.ats"

[Nest FilePath] – This outputs the entire file path to the ToolShop nest file.

Eg. "C:\WorkFiles\ToolShopNests\Example_Nest.ats"

3. Reduce Human Error using Nest Labels

Nest Labels are another great feature within ToolShop that act as an extension to Nest Reports.

Upon exporting from ToolShop, a Nest Label sticker may be printed and applied to sheets of material that are waiting to be cut. As the operator loads the next sheet in his stack, he can easily use the attached Nest Label to either scan the filename using a barcode reader, or manually enter the filename on the label.

Just like using a Nest Report, operators are easily able to:

- Identify that they have loaded the correct file using the nest representation
- Verify the tooling to be used that are listed on the label
- Verify the correct material will be used

Nest Labels have the same fields and symbols as those available on a Nest Report.

Through the use of Nest Labels you can improve efficiency and reduce human error.

4. Track Stock with Sheet Labels


Beginning with ToolShop v2, the material management library has been designed so that businesses can easily track their material stock levels.

In the material management library, each material type and remnant are given a unique ID. This means that if there are 20 sheets of the same material, all with the same size and thickness, all these sheets will have a single unique ID. On the other hand, if a remnant is created, it will receive a new ID specific for that individual remnant.

When setting up your material stock in ToolShop, you may then print a Sheet Label to place on each pallet of sheet material. This will allow the operator to match the material ID from a Nest Label/Report with the stocked material or remnant to use. This can significantly reduce cases of using the incorrect material, especially when different grades of material may be available in the same sizes.

There are two stages at which you would print Sheet Labels.

Printing sheet labels at time of receiving stock

If you are just setting up ToolShop or if a new material is added to the material library, Sheet Labels can be printed for these new materials. Within the Material Library, simply select a single row, or select multiple rows by holding CTRL while selecting. Then click the  button to print these labels.

Also, if a remnant is added to the stock, a single remnant Sheet Label could also be printed for this. Although, usually this would be done at the time of exporting as we will discuss now.

Printing Sheet Labels at time of export

Sheet Labels may come in handy when exporting a job. The common process of using Sheet Labels is to use them in combination with Nest labels. Nest Labels are printed with the export of each sheet. Using the Sheet ID you placed onto the Nest Label, you can easily match it the already printed Sheet Labels on stocked material pallets.

If you choose to do so, when ToolShop exports a job, you may allow it to automatically create any remnants and add them to your material library. In the ToolShop export dialog, you can set it to automatically print a Sheet Label for the newly created remnant. This makes management of remnants easy and effective.

Symbols

Below is a description of the special symbols available on Sheet Labels.



Sheet Shape

The sheet symbol is an effective way of identifying sheets by its height/width ratio and also if its an oddly shaped remnant. The Sheet Shape symbol will print an outline of the sheet.

Fields

Below are the fields specific for Sheet Labels. To see all the generic fields available see Section 6: Using the Designer.

[Is Remnant?] – This prints either ‘Yes’ or ‘No’ depending on whether the target sheet is a remnant.

5. Part Labels

Part Labels are an incredibly simple feature that any profile cutting business should take advantage of. Through the use of part labels, it's possible to easily and efficiently track parts all the way from a DXF through to the final cut part.

Part labels are printed for every ‘group’ of shapes in your ToolShop nest on specific layers. Once the labels are printed and the job has been cut, using the Part Location Indicator and the Shape Preview, the operator can apply these labels onto the cut parts. Part labels will be in order from bottom to top, left to right (columns).

Why use part labels?

The information that can be printed on one of these labels can allow you to track the individual destination of each part, from a single template using these values.

Examples of using Part Labels Effectively

For example, a kitchen cabinet maker has two customers' jobs on one nest. Once cut, the job is essentially a collection of specifically sized panels. Using Part Labels printed from ToolShop, the operator can apply one label per panel with the name of the panel, which direction was originally north (up) using the Part Compass, the customer's details and also any other 'custom properties' added to the shapes.

A second example, a business does contract cutting. They have a large nest with possibly hundreds of parts from a range of customers. Trying to identify each part after they have been cut is extremely time consuming. Using the 'Custom Properties' of ToolShop, as they import files into the nest, they add the details of the customer including the shipping address, business name, phone number, etc. When the job is cut, they may now simply go from bottom to top, left to right applying the stickers by matching the Shapes Preview and Location Indicator to the one just cut. These parts can then be sent directly to the customers with a minimum of effort in trying to identify the individual part owners.

Tell ToolShop to Print a Label for a Part

In order for ToolShop to print a label for a part, it's required to mark which layers to find parts. ToolShop finds parts needing a label from set layers, and marks anything grouped with it as the same part.

You may mark your shapes main layer as a label layer and simply group all other shapes within the part to the main outer shape.

If though you need higher control over what gets labels, you may set a specific layer just for label placeholders. Settings the Custom Properties of this guarantees the shape in which the Custom Properties are retrieved, and also, there is no need to manage multiple shapes. See the illustration.

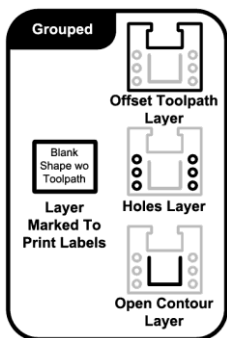


Figure: You may set specific layers as label placeholders.

In order to set which layers are marked Part Label layers, go into *File Menu->Label Setup->Design Part Label Template...* and then click the menu at the top saying "Set Label Layers". You will then be presented with a list of layers with tick boxes. Ticking a layer name will assign that as a layer to find parts on.

Symbols

Below is a description of the special symbols available on a Nest Report.



Shape Preview

The Shape Preview gives operators the ability to identify at a glance, the correct part for the Part Label. A scaled preview of the shape is printed onto the label to fit the box that was placed on the label designer.



Part Compass

The Part Compass on a label applied horizontally onto the nest will point in the direction of the parts original

north. For example, if a part after being imported was rotated 45° degrees clockwise, the compass would point north-east.



Nest Indicator

As parts can often be very similar, the nest indicator gives you a preview of the entire nest, with the labels part filled with black. This makes identifying an area of similar parts a breeze.

Custom Property Fields

Part Labels have the ability to print the Custom Properties provided by a shape. This allows you to have a lot more information on each Part Label. Each of the Custom Properties within a shape will automatically appear in the “Select Fields...” dialog in the designer. These values will have arrow brackets (<>) surrounding them.

Fields

Below are the fields specific for Sheet Labels. To see all the generic fields available see [6. Using the Designer](#).

[Part Filename] – The filename of the ToolShop ATS file.

[Bounds Width] – The maximum width of the rectangle bounds.

[Bounds Height] – The maximum height of the rectangle bounds.

[Bounds Area] – The area used by the rectangle bounds.

[Part Sheet #] – The sheet number the target part is on. (Usually 1, 2, 3, 4...)

6. Using the designer

You may note that ToolShop uses a similar designer for all print documents. The difference is that each of the designers may have slight variations in the items available, as well as variations in the way they're output.

First, open the specific designer required for the type of print document. The designers are usually located within the “*Printing Menu*”. With some documents you may want a larger sheet of paper, if so then it may be easier if you make the window full screen.

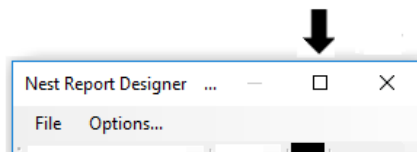
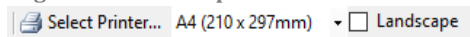


Figure: Switching to the full screen mode can be helpful especially when designing nest reports.

Page & Printer Setup



In order to do any printing, the paper or label size and printer need to be selected. To change the printer, click the ‘Select Printer...’ button in the main toolbar of the designer and choose the desired printer using that window.

Changing the paper or label size can be done through the ‘Select Printer’ window or through the dropdown box beside the button. Toggling the landscape checkbox alternates the orientation between portrait and landscape.

Symbols

Below is a list of the basic symbols available across all print documents.

A Text

Text symbols allow you to write descriptive labels onto the document. Simply clicking a text symbol allows you to edit its text.

{P} Fields

Fields are just like Text symbols except what is written in the designer will not be printed; instead the value of the field's description will be printed. When placing a field, the field selector window is displayed. This gives you a list of available fields that can be used.

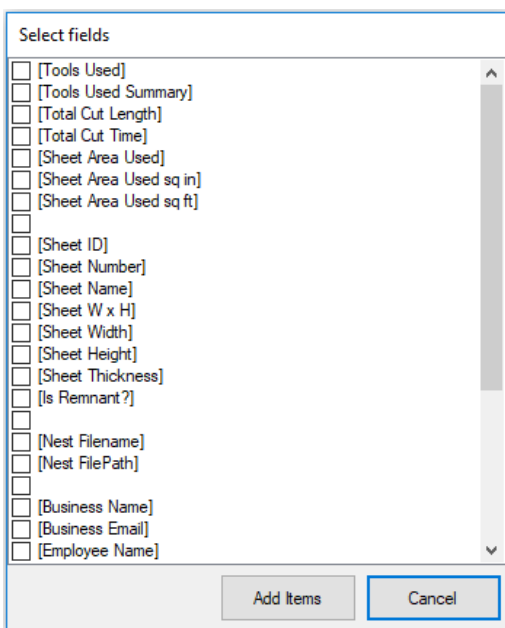


Figure: Switching to the full screen mode can be helpful especially when designing nest reports.

Barcode

Barcodes are exactly like fields in the way they work. Adding a barcode will ask what format to use and then show the Select Field window. Once the barcode has been added to the designer, it will simply display a dummy barcode with the value 12345. Upon printing, it will print a barcode with the same result that a Field symbol would.

Image

Clicking on the add image button will show a 'File Open' window. Use this window to navigate to any jpg, bmp, png or gif. It will then immediately add it to the designer.

Fields

[Sheet ID] – This is the unique material ID of the sheet this print document is referencing.

[Sheet Name] – The name of the material this sheet uses. Eg. Aluminium, Steel, Acrylic, MDF.

[Sheet W X H] – The width and height of the sheet.

[Sheet Width] – The width of the sheet along the X axis.

[Sheet Height] – The height of the sheet along the Y axis.

[Sheet Thickness] – The thickness of the sheet along the Z axis.

[Business Name] – Set this in ToolShops preferences.

[Business Email] – Set this in ToolShops preferences.

[Employee Name] - Set this in ToolShops preferences.

[Business Fax] - Set this in ToolShops preferences.

[Business Phone] - Set this in ToolShops preferences.

[Business Postal Address] - Set this in ToolShops preferences.

[Business Website] - Set this in ToolShops preferences.

[Business Email] - Set this in ToolShops preferences.

[Date DD/MM/YYYY] – Todays date formatted into “Day/Month/Year”

[Date MM/DD/YYYY] – The date and time formatted into “Month/Day/Year”

[Date DD/MM/YYYY HH:MM:SS] – The date/time formatted into “Day/Month/Year Hours:Minutes:Seconds”

[Date MM/DD/YYYY HH:MM:SS] – The date/time formatted into “Month/Day/Year Hours:Minutes:Seconds”

Designing a Label

Designing a label is very simple. Along the left hand side of the label designer, there are buttons to add various symbols or fields. Each label type has its own specific symbols to use and have been listed along with each label type, along with the generic ones that are directly above.

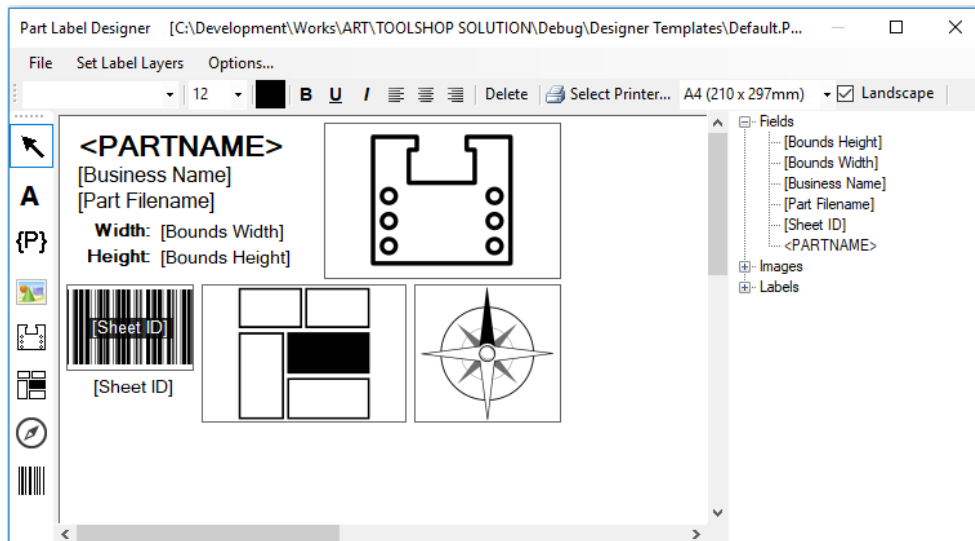


Figure: Part Label Designer main window.

When clicking one of the buttons, a new symbol will be added to the main designer window in the top left corner.



ART ToolShop

To select a symbol or field, simply click it.

To move, simply click and drag from the centre of the symbol.

To resize, use the black square nodes on the edges of the symbol to resize.

To delete, simply press the delete key on the keyboard, or click the delete button on the top menu.

Text Fields

Text fields are able to have their fonts changed, font size changed, made bold, italic and underlined, as well as have left, right and central justification. These controls are located on the top menu bar.