Torchmate CAD Version 7.1 User's Guide

Revised: October 2008

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Overview of this Manual:

Torchmate CAD is an easy to use **C**omputer **A**ided **D**esign software package designed for use with Torchmate CNC systems. Torchmate CAD includes many features that are designed for plasma cutting including nesting, DXF importing, kerf compensation, cut order, lead in / out creation, and scanned image editing just to name a few. This manual gives a general overview of what Torchmate CAD's capabilities are. When teaching a computer program that is designed to create almost anything that can be imagined there is a certain amount of information that will inevitably be left out. It is impossible and pretentious to assume that a manual could encompass the entire scope of every item that could be created with Torchmate CAD. Likewise for some processes or designs there can be multiple ways to complete the same goal, in cases like this each way may be presented. This manual is a growing walkthrough that started with questions from phone and email, led to simple walkthroughs, and now a cohesive document that outlines the operation of Torchmate CAD. As such any suggestions for new methods, sections, or design ideas are always welcome. Remember Torchmate technical support is free and unlimited if there are any questions that arise from this manual please do not hesitate to contact us. Thank you for purchasing Torchmate CAD, welcome to the world of CNC tables and computer design.

Torchmate Technical Support

866-571-1066

775-673-2200

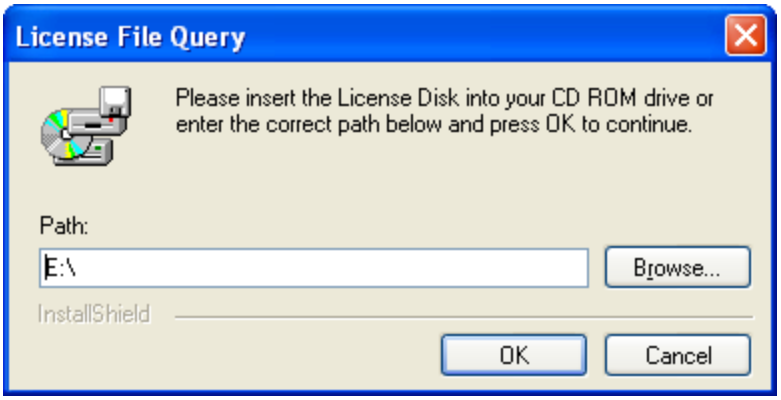
support@torchmate.com

Installing Torchmate CAD:

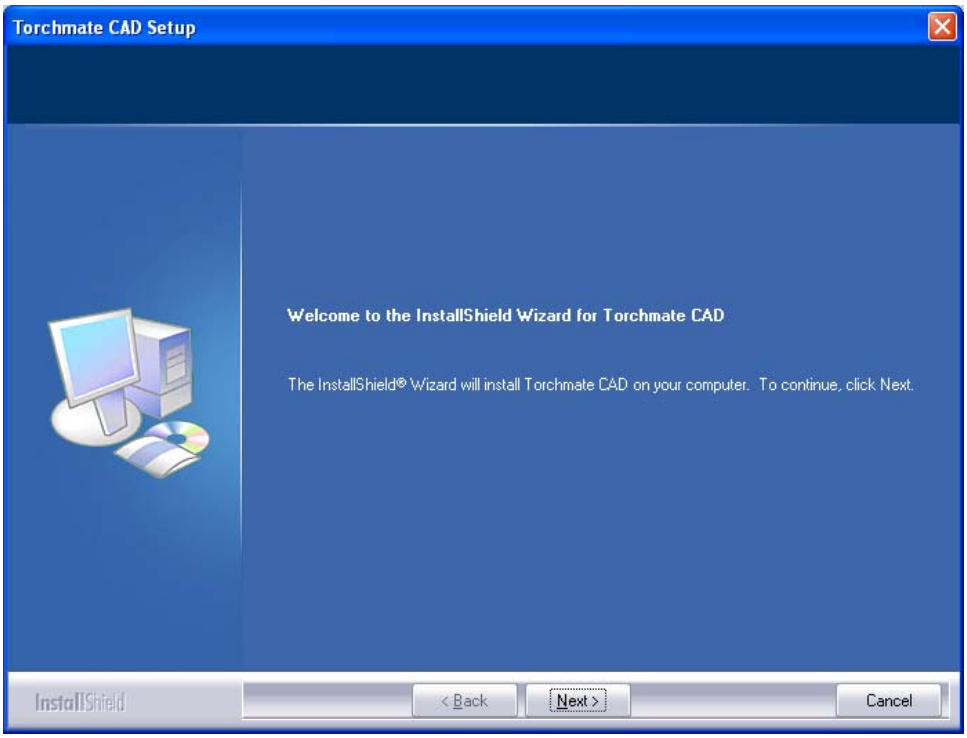
To begin installation of Torchmate CAD insert the Torchmate CAD CD packaged with a security dongle. The installer will automatically launch. The first prompt will be to select the setup language. This language is only for the installer, Torchmate CAD menus are only in English.

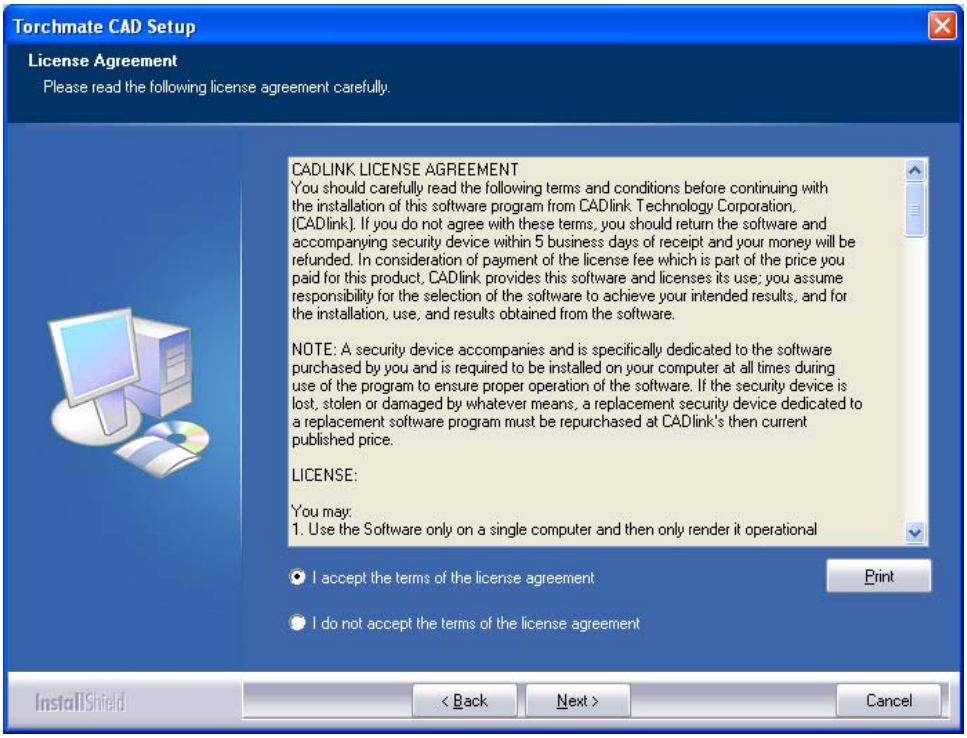
After selecting a language the installer will issue a reminder about inserting the security dongle. Follow the instructions for the specific dongle being used.

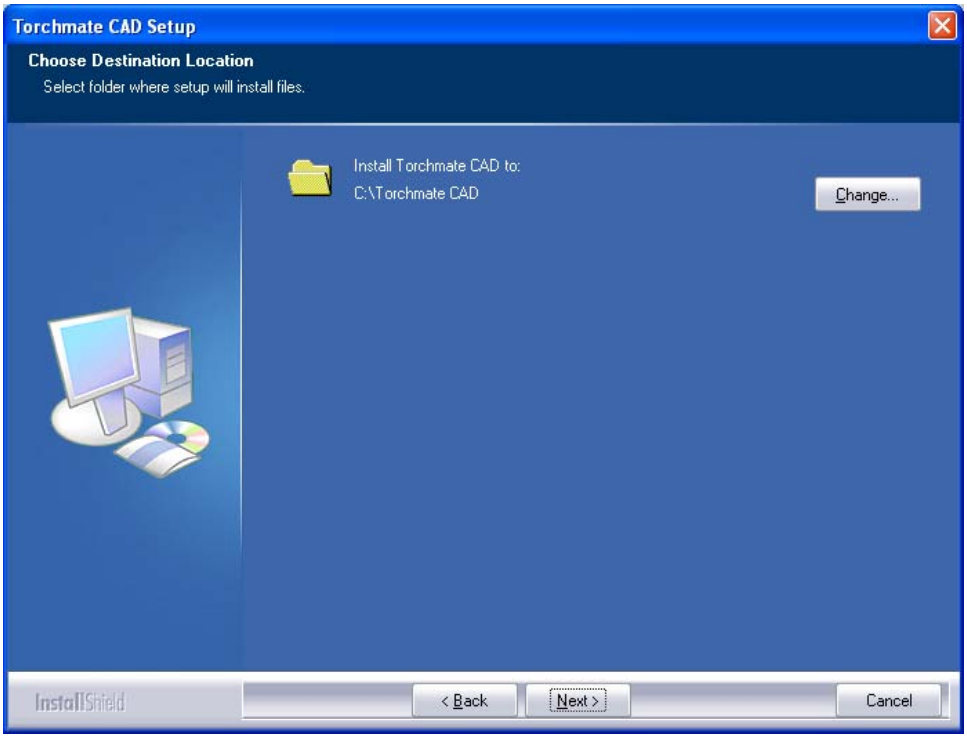
If a ‘License File Query’ screen appears the dongle has not been read yet or was inserted too soon. If using an orange dongle the license files will be on the dongle which will appear as a removable media. If a blue or purple dongle is being used the license files will be on the installation CD.



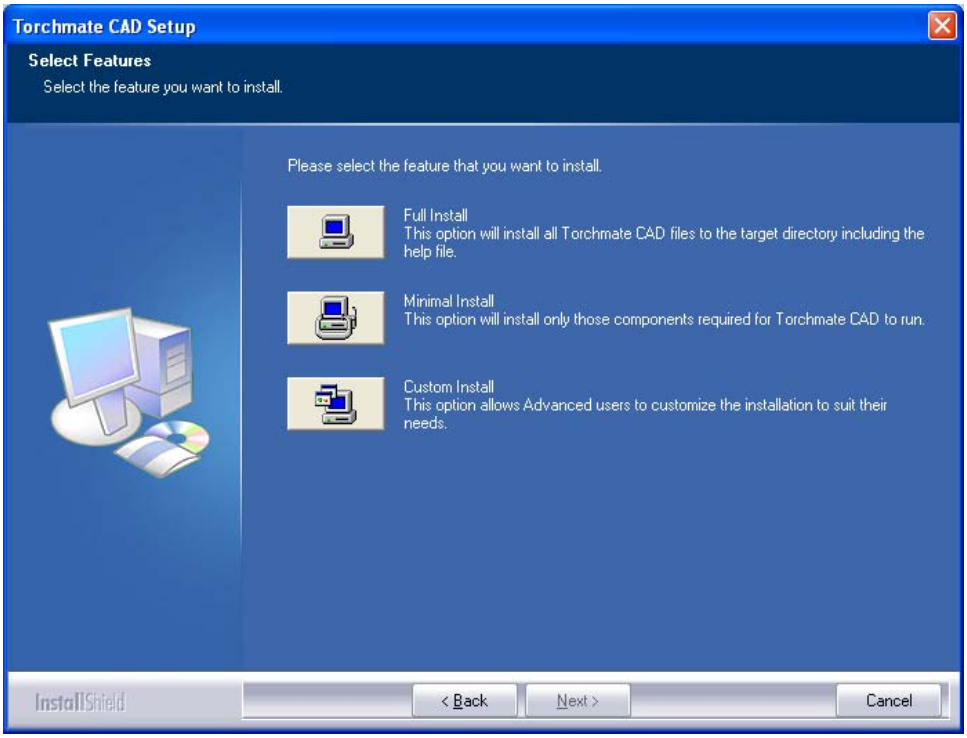
Continue with the installation.

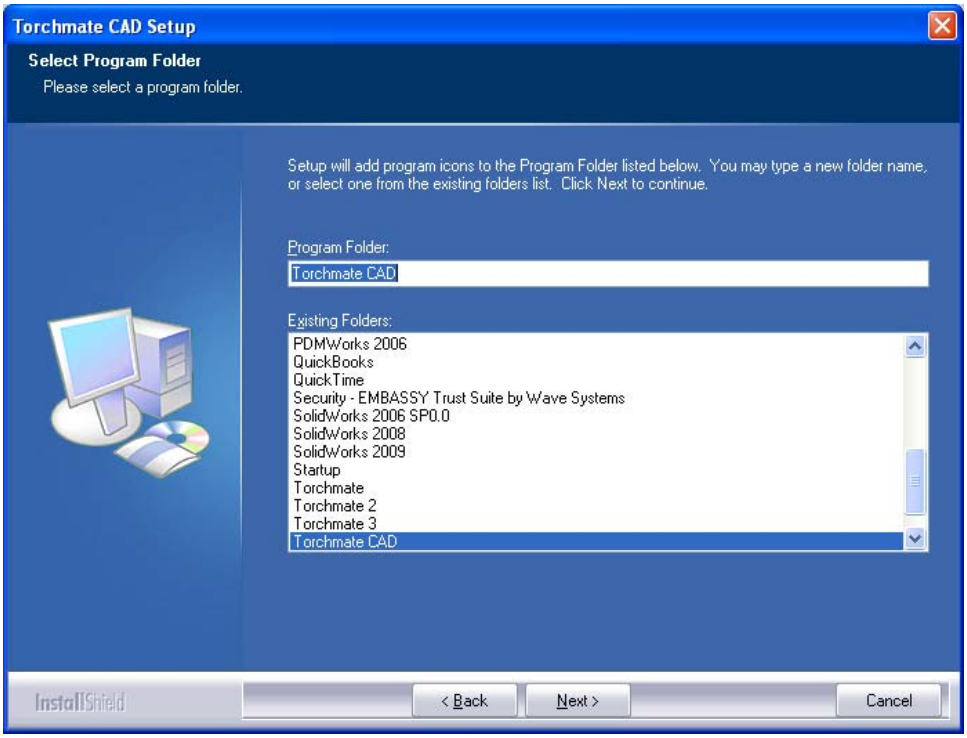


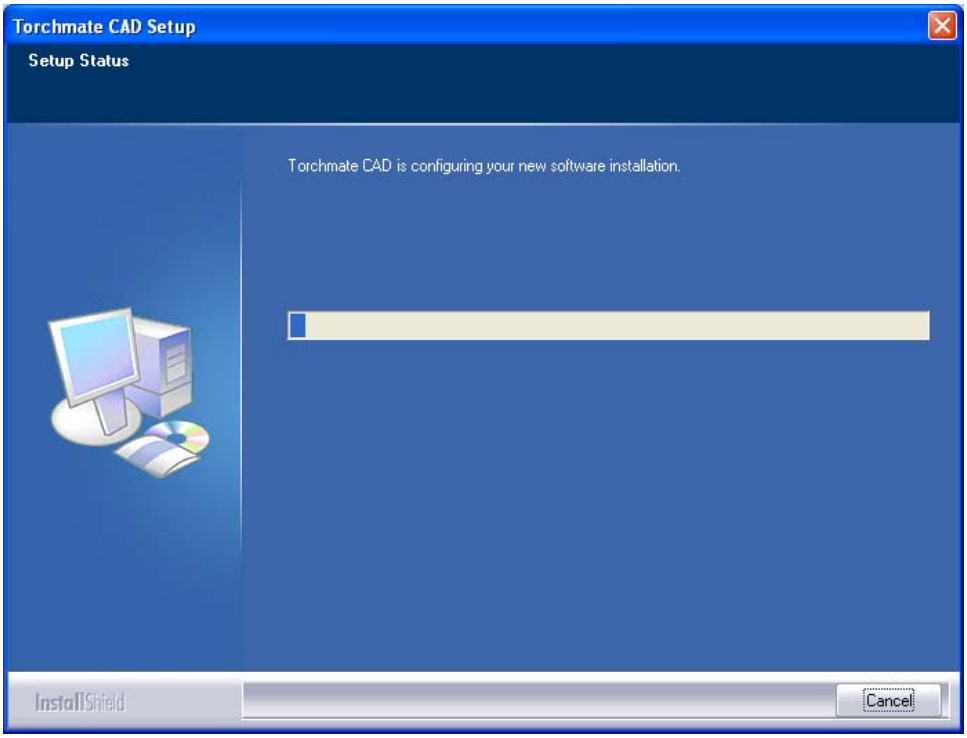

Torchmate CAD Setup
License Agreement
Please read the following license agreement carefully.
CADLINK LICENSE AGREEMENT
You should carefully read the following terms and conditions before continuing with the installation of this software program from CADlink Technology Corporation, (CADlink). If you do not agree with these terms, you should return the software and accompanying security device within 5 business days of receipt and your money will be refunded. In consideration of payment of the license fee which is part of the price you paid for this product, CADlink provides this software and licenses its use; you assume responsibility for the selection of the software to achieve your intended results, and for the installation, use, and results obtained from the software.
NOTE: A security device accompanies and is specifically dedicated to the software purchased by you and is required to be installed on your computer at all times during use of the program to ensure proper operation of the software. If the security device is lost, stolen or damaged by whatever means, a replacement security device dedicated to a replacement software program must be repurchased at CADlink's then current published price.
LICENSE:
You may:
1. Use the Software only on a single computer and then only render it operational
I accept the terms of the license agreement
Print
I do not accept the terms of the license agreement
InstallShield
< Back
Next >
Cancel


Torchmate CAD Setup
Choose Destination Location
Select folder where setup will install files.
Install Torchmate CAD to:
C:\Torchmate CAD
Change...
InstallShield
< Back
Next >
Cancel

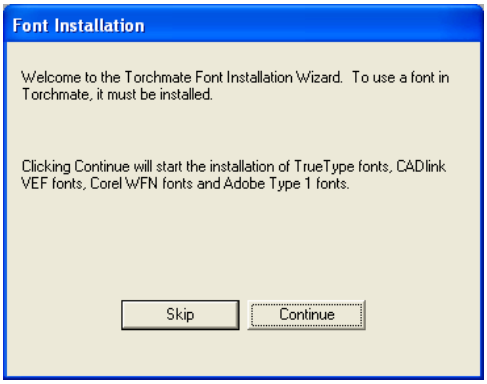
When prompted select ‘Full Install’.

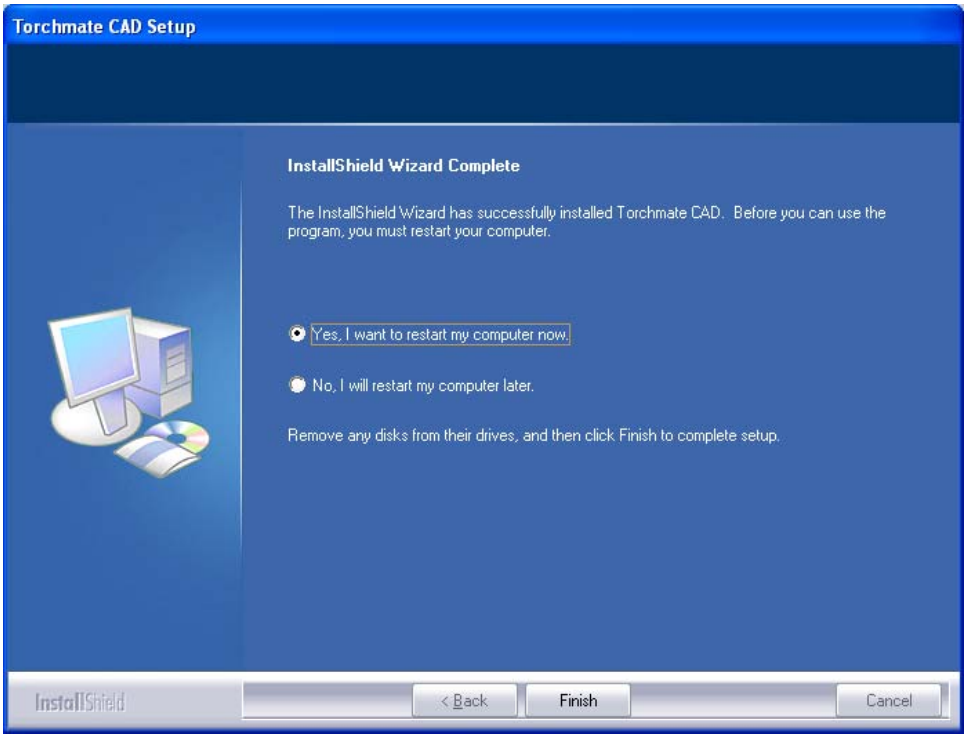






After the main CAD program is installed the installer will want to search for all fonts installed on the computer. This is not necessary to run Torchmate CAD but allows all fonts installed on the computer to be accessed by Torchmate CAD.

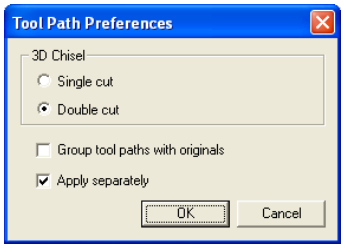




Torchmate CAD is now ready for use; if a purple or blue security dongle is being used insert it after the computer has restarted.

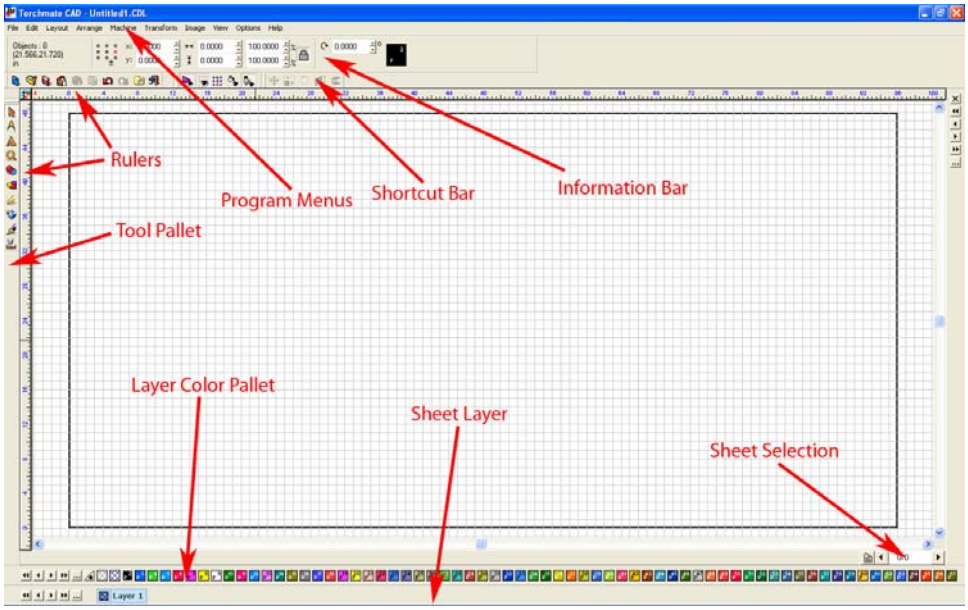
****IMPORTANT CONFIGURATION CHANGE****

It is necessary to change one default setting to allow simpler tool path creation. Go to 'Options'→ 'Torchmate Setup' → 'Tool Path Preferences'. Uncheck the option to 'Group tool paths with originals'.



Overview of Menus:

All functions of Torchmate CAD are accessed from the main screen. The largest feature of the main screen is the work area; this section contains a gridded work area that is skirted by a material blank. The main work area is where all drawing will be done. Around the work area are various tool bars that contain all of the tools, shortcuts, and options for creating a drawing.



Rulers: The rulers sit on either side of the work area and provide a reference to the coordinates of a spot in the work area.

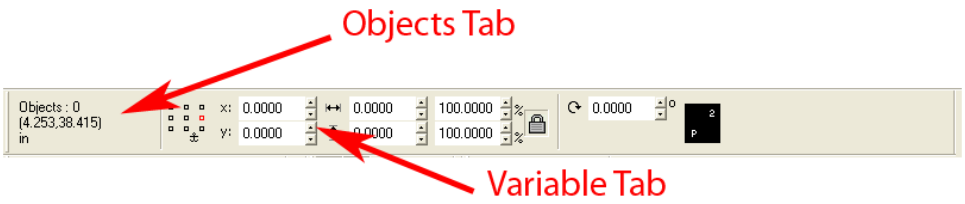
Layer Color Pallet: This bar allows the color of an object to be changed; this is useful in keeping track of various parts. The color of an object does not affect the cutting process at all and in most cases this is an unused tab. Colors can be changed by double clicking on one of the colors, this opens a window to change the color options.

Sheet Layer: The sheet layer is a guide used to differentiate multiple sheets of multiple objects. This bar like the 'Layer Color Pallet' does not affect the cutting process and is generally unused. The sheet layer is visible when 'Show Fill' is enabled in the 'View' menu, by default it is transparent.

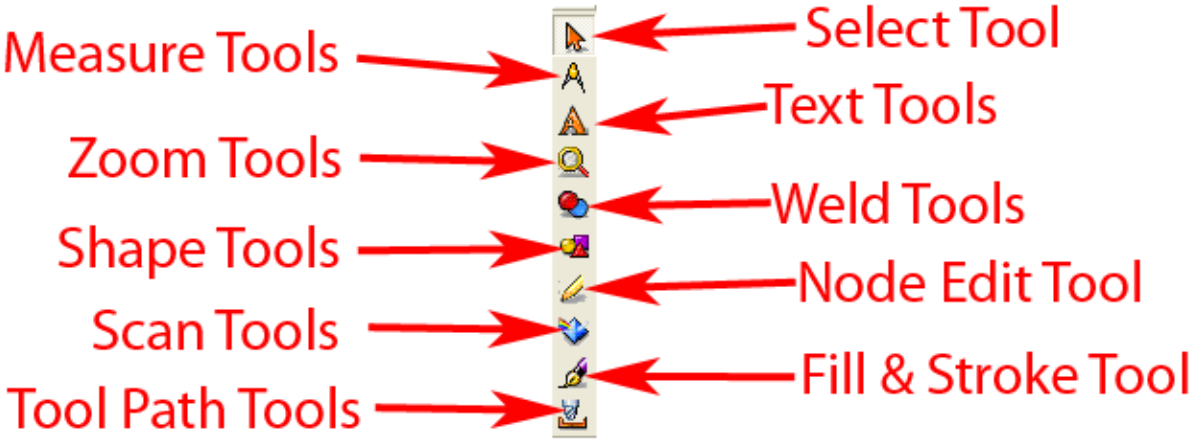
Sheet Selection: This option is used to toggle between multiple sheets. Multiple sheets occur when nesting or other options that will create additional sheets. This option enables these sheets to be paged through if multiple sheets exist.

Shortcut Bar: The shortcut bar contains icons that serve the same function as certain options in the program menus. Generally these options are for selections that are used frequently. This menu can be customized by going to 'View' → 'Toolbars' → 'Customize…' various functions can be added or removed from the bar.

Information Bar: The information bar contains basic information about a selected object. This bar contains two main areas, the objects tab and the variable tab. The objects tab gives two bits of information, the coordinates of the pointer in the grid and how many and what type of objects are selected. This bar is useful when determining what is selected and also if what is selected is the correct object. The variable tab is a dynamic bar that will change with each tool and part selected. This bar displays options for various tools when they are selected.



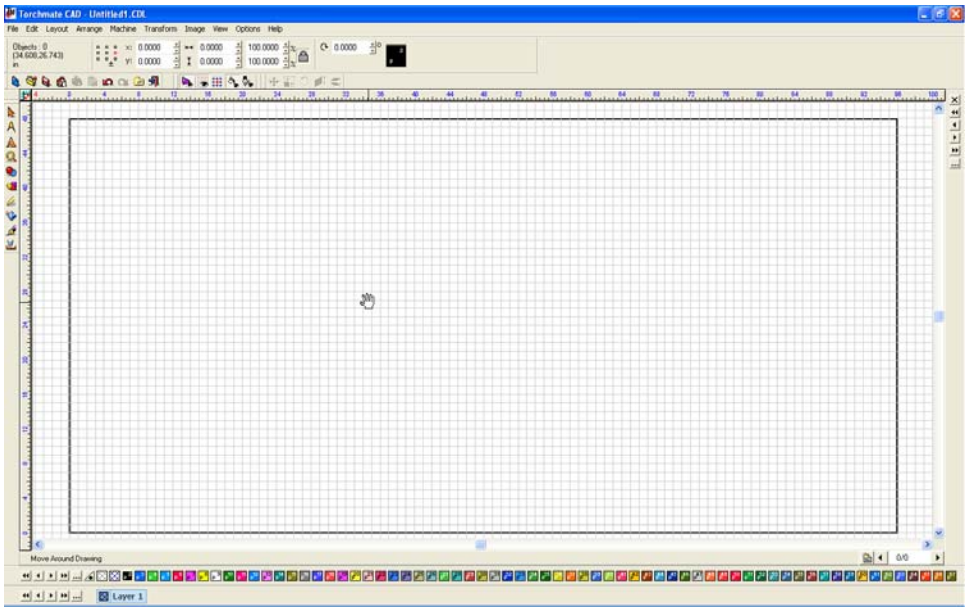
Tool Pallet: The bar on the left hand side of Torchmate CAD contains the pull out menus for basic drawing tools as well as the means to manipulate them. Each tool will be further explained throughout this manual but essentially any drawing from scratch will start here.



Navigation and Selection Methods:

Being able to move around the drawing area and manipulate objects is the key to successful drawing. In Torchmate CAD there are a few ways to simplify this process so that drawing can be done more efficiently.

Moving around the work area is done in a few ways. The scroll bars along the bottom and right of the area can be used to pan the page. Another way is to use click the third mouse button this brings up a pan hand that allows click and drag movement. On most computer mice the third mouse button will activate when the scroll wheel is depressed.



The pan tool can also be accessed in the 'Zoom Tools' menu. Also in the 'Zoom Tools' menu are ways to either zoom in or out. There are short cuts to all of these or they can be accessed from the menu.

Zoom Shortcuts:

Zoom In: F5

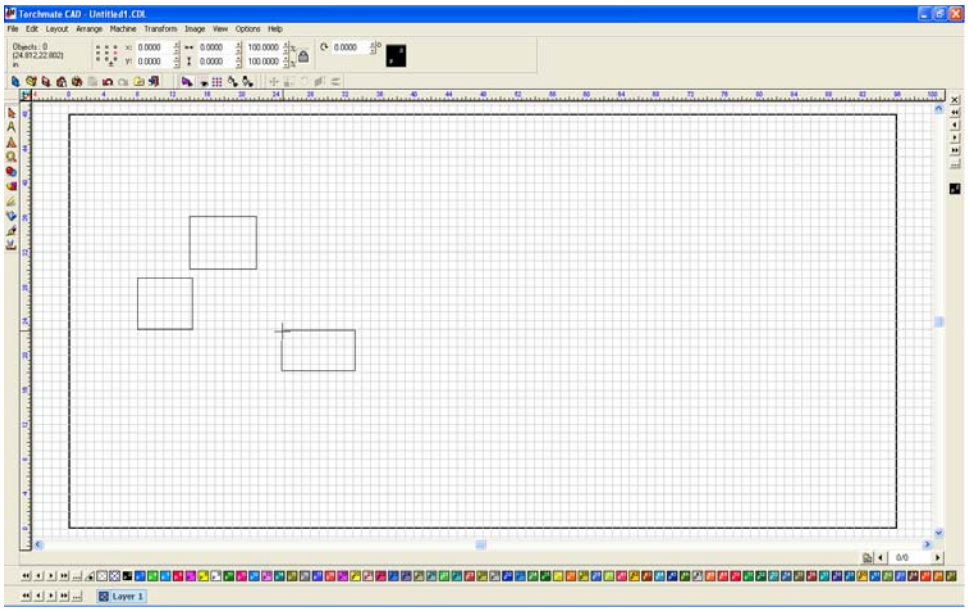
Zoom Out: F6

Zoom to selected Object: F7

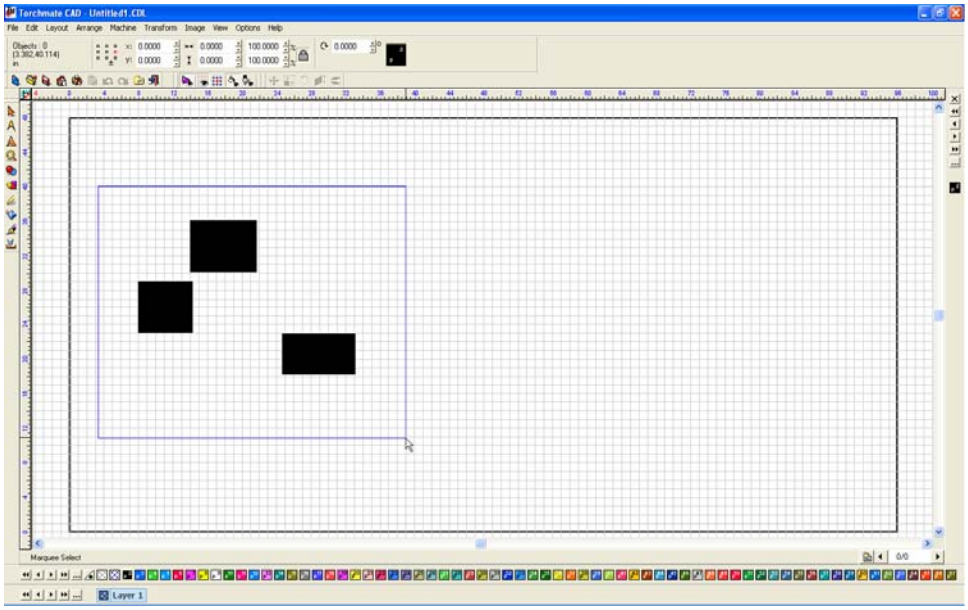
Zoom to Material: F8

Pan State: 3rd Mouse Button

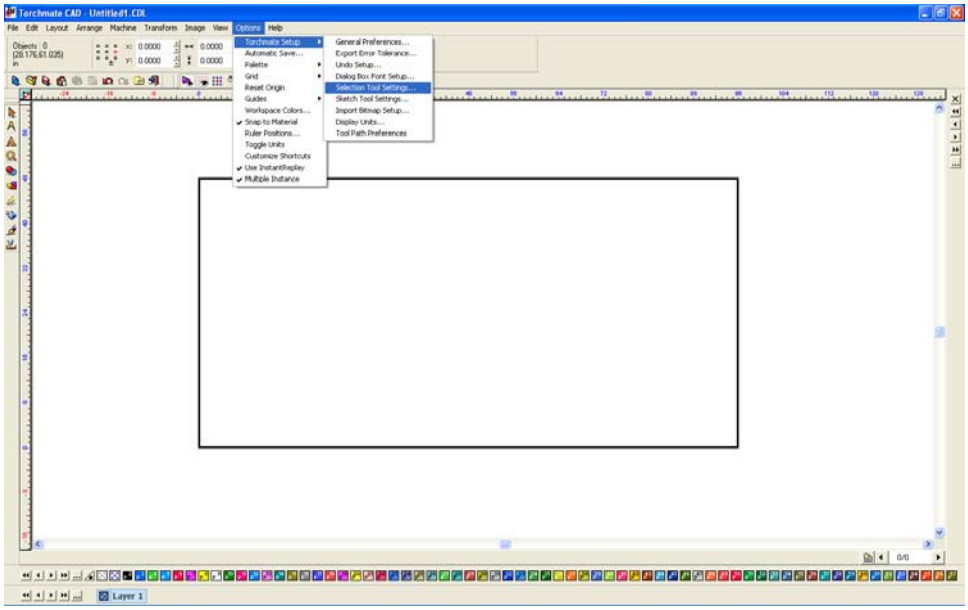
Selection of objects can be crucial for certain drawing methods in Torchmate CAD. There are multiple ways to select an object. With pressing the 'TAB' key will select one object, pressing 'TAB' repeatedly will cycle through each object generally in the order they were created. A single object can also be selected simply by clicking on the edge of the object. To ensure that the object is gets selected the mouse will change into a crosshair when it is over the edge.



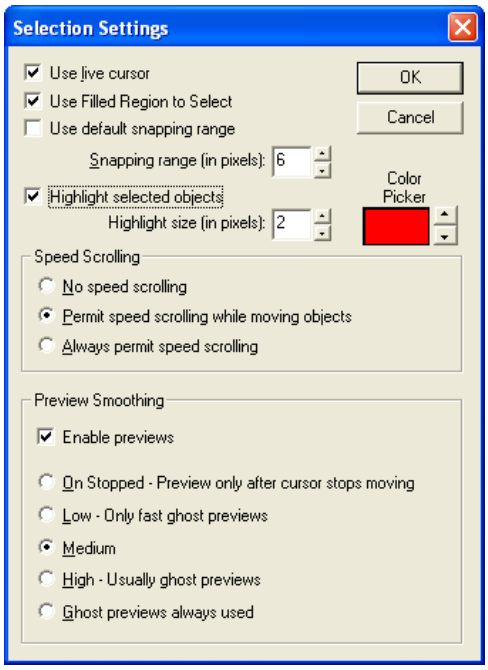
Multiple objects can be selected as well, the simplest way to do this is to click and drag with the 'Selection Tool' around the objects that are to be selected. Multiple objects can also be selected by using the 'Shift' key. With this method begin by selecting one object then hold down 'Shift' on the keyboard; click on another object both objects will now be selected. This method also works in reverse for unselecting a single object when multiple objects are selected.



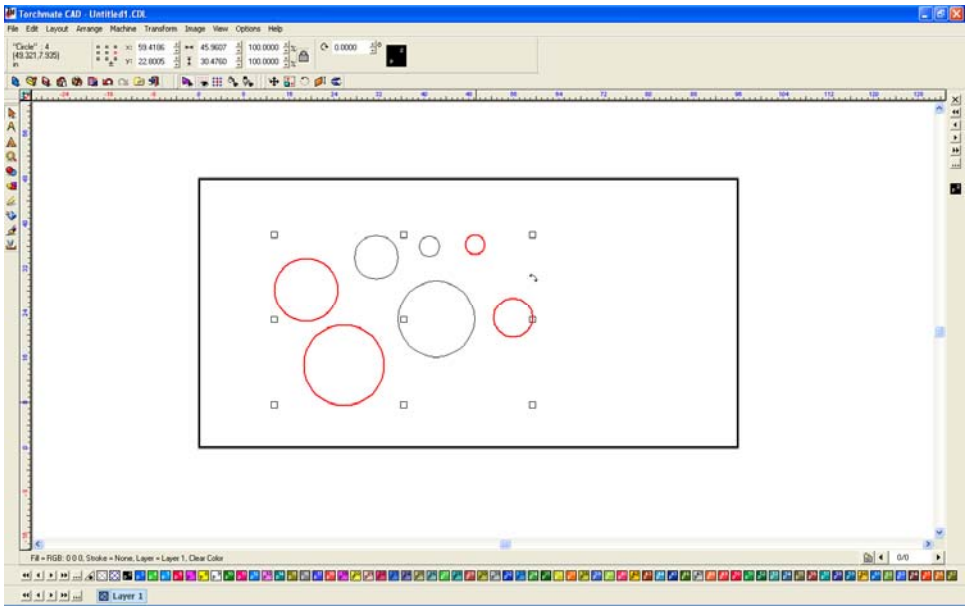
When selecting multiple objects it can sometimes become ambiguous as to which ones are selected and which are not. In Torchmate CAD there is a feature that when enabled highlights each selected object a certain color. This allows for easy identification of selected objects. To enable this feature go to 'Options' →'Torchmate Setup' →'Selection Tool Settings'.



Check the box labeled 'Highlight selected objects' the 'Color picker next to this option adjusts the color.



With this feature enabled any selected object will have a colored highlight around it for easy identification.

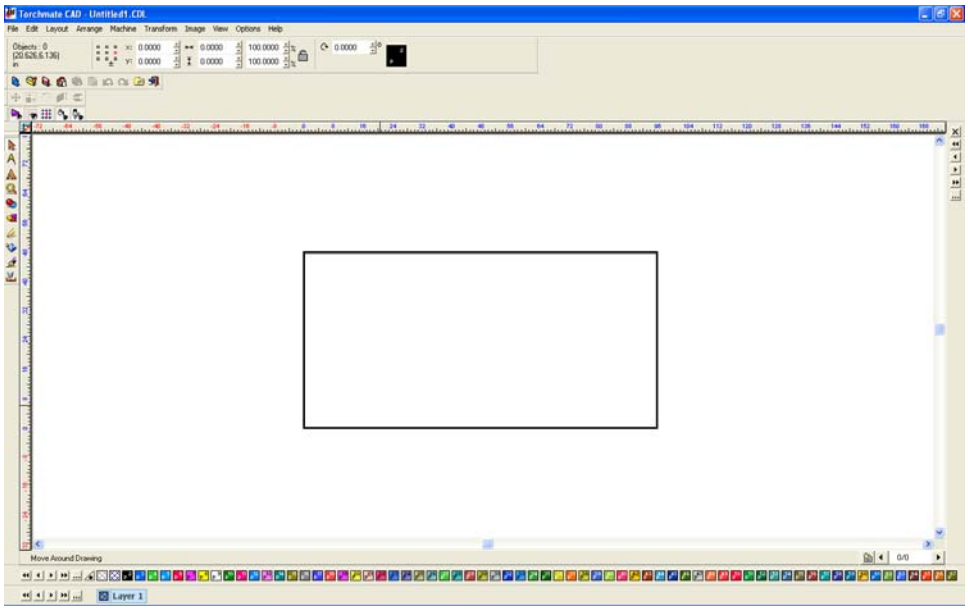


Simple CAD Drawing:

This section describes basic drawing of a simple shape and how to prepare this shape to be cut. Please refer to later sections of this manual for further explanation on topics discussed.

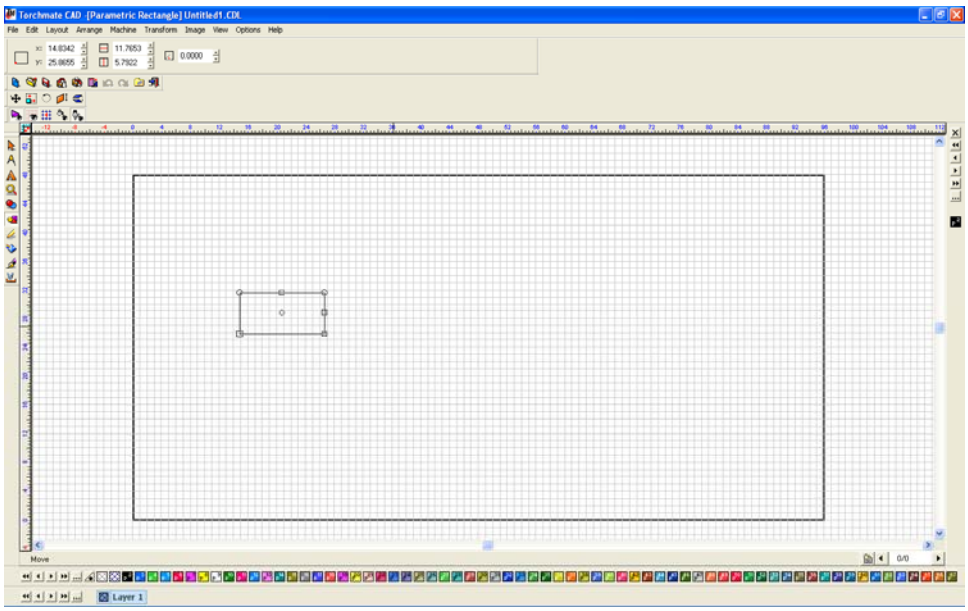
Creating a Shape:

The Torchmate CAD's main screen has a tool bar along the left-hand side of the screen and an information bar along the top. All drawing is done in the main screen defined by the rulers along the margins. The top bar displays the size of the object, its position and scale. The left-hand tool bar contains tools used for drawing and manipulating objects.

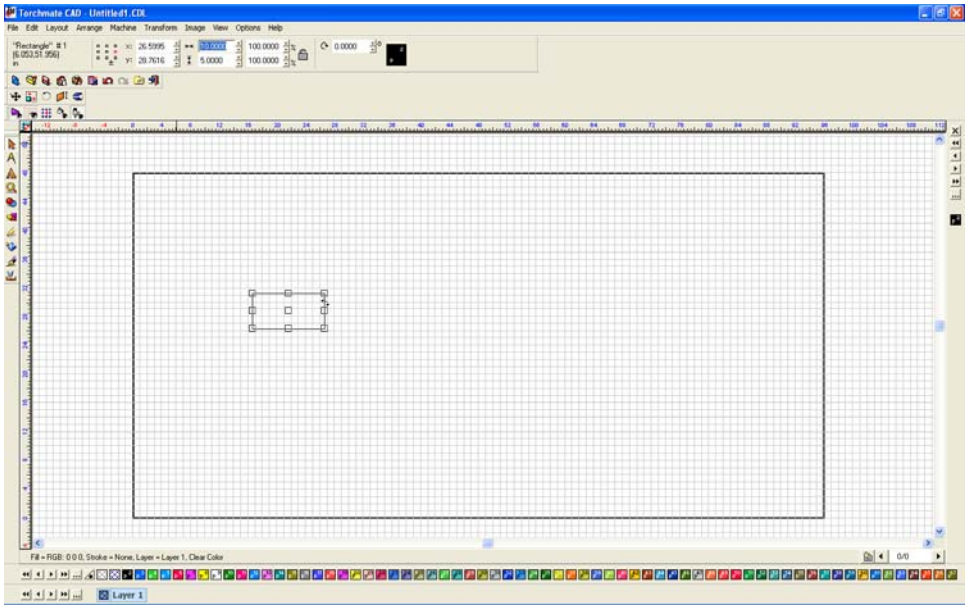


On the left-hand bar the shape tools can be accessed. The 'Shape Tools' menu has a few simple shapes to start most part drawings with (circle, oval, square, polygon, star, arrow and arch).

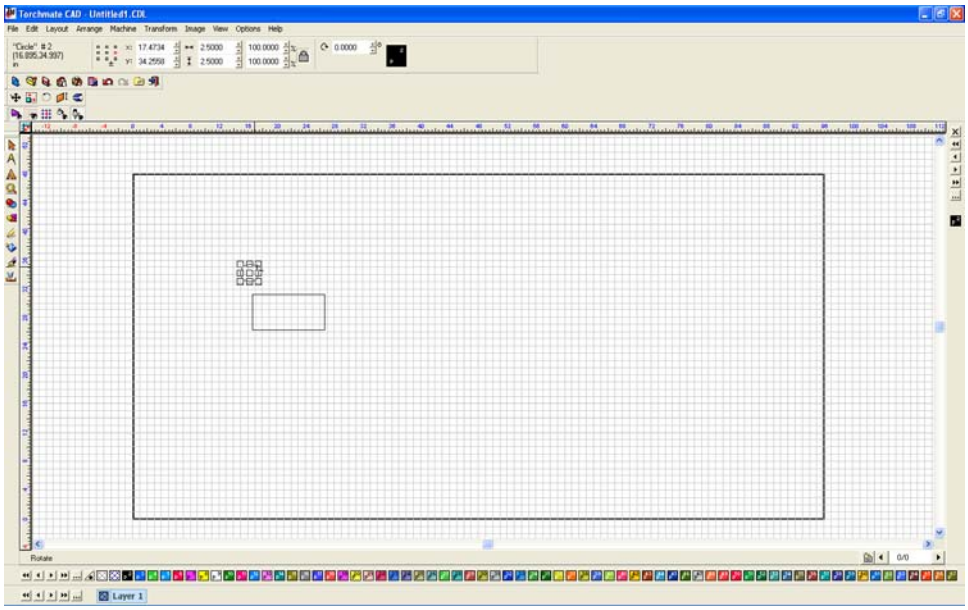
Begin by drawing a simple rectangle; the size is irrelevant as it can be adjusted later. To do this simply click on the rectangle icon from the 'Shape Tools' menu, then click and drag to draw out the rectangle in the work area.



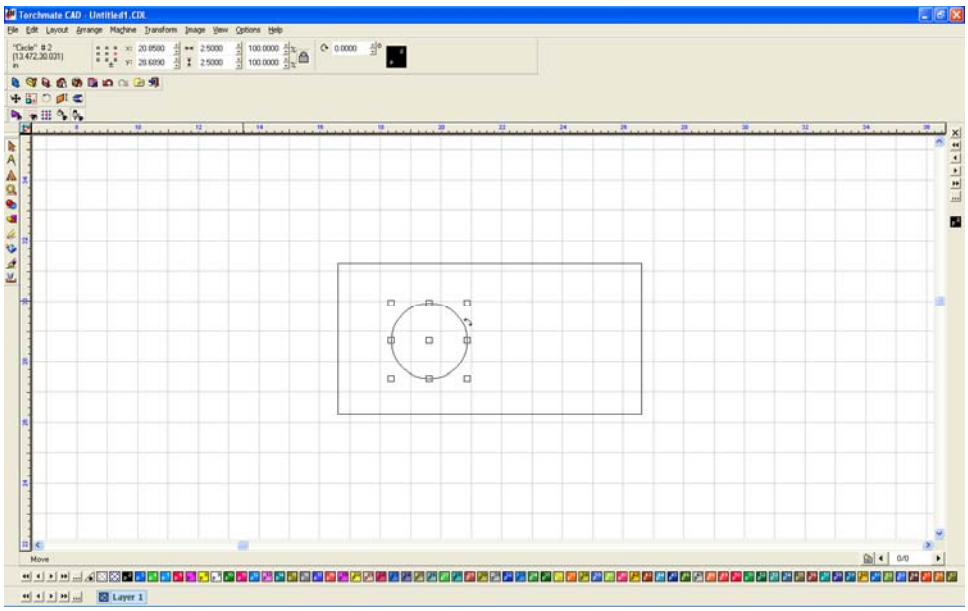
To change the rectangle's size select it by clicking on the edge of the part, then on the top tool bar type in the 'X' and 'Y' dimensions that are wanted and press enter.



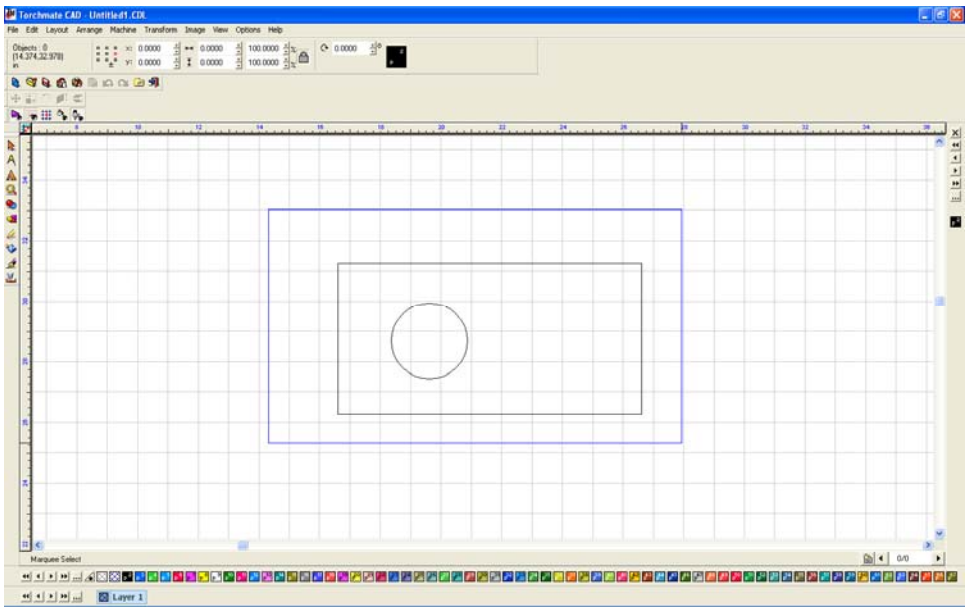
Draw a circle using the same method.



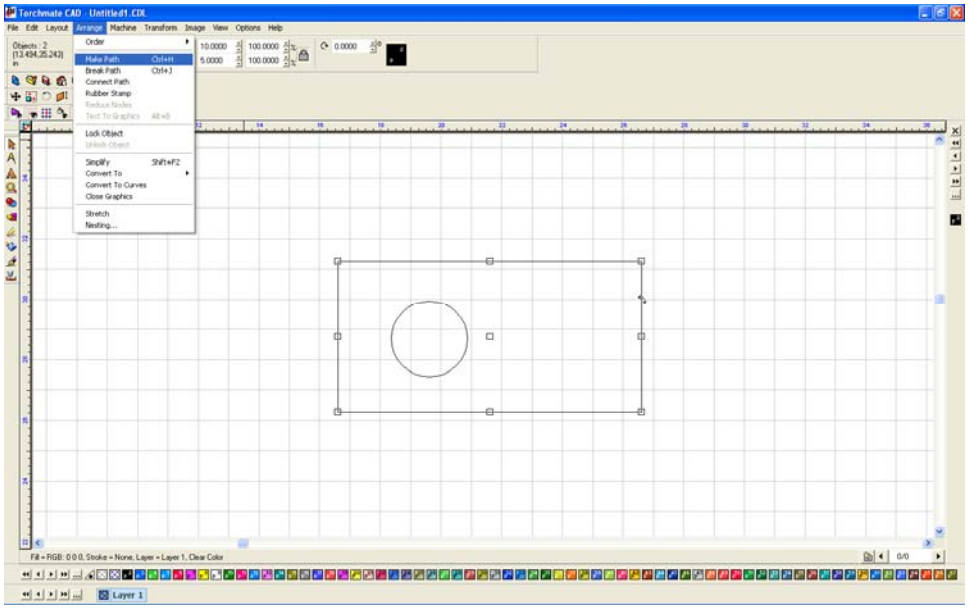
Zoom in on the shape by using the ‘Zoom Tools’ pull out along the left-hand tool bar. Next move the circle to position it inside of the rectangle.



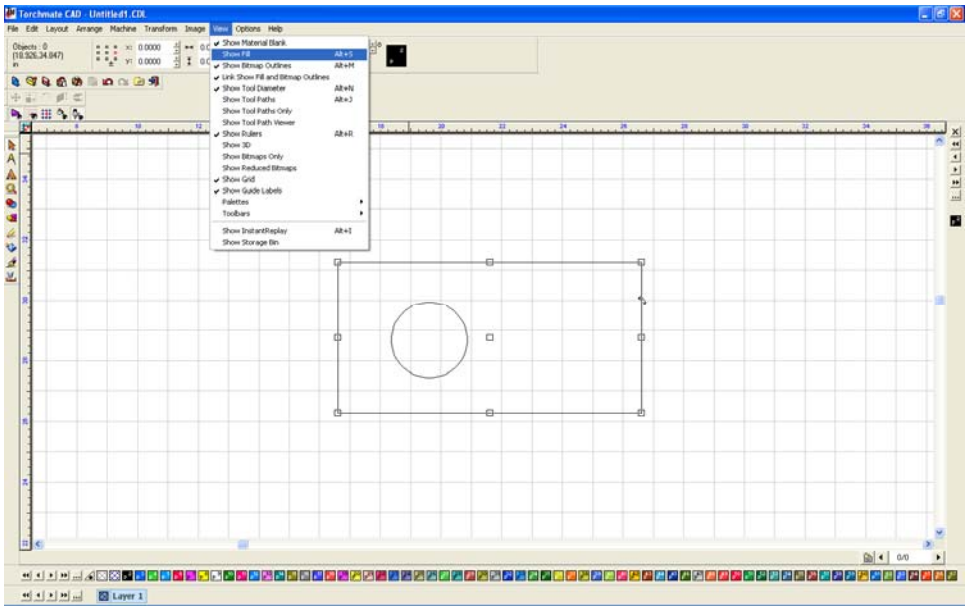
Once the circle is positioned in the rectangle select both objects, this can be done a few ways. Using the ‘Selection Tools’ from the left-hand toolbar click and drag a blue box around the two objects this will select both of them; another way is to go to ‘Edit’ → ‘Select All’.



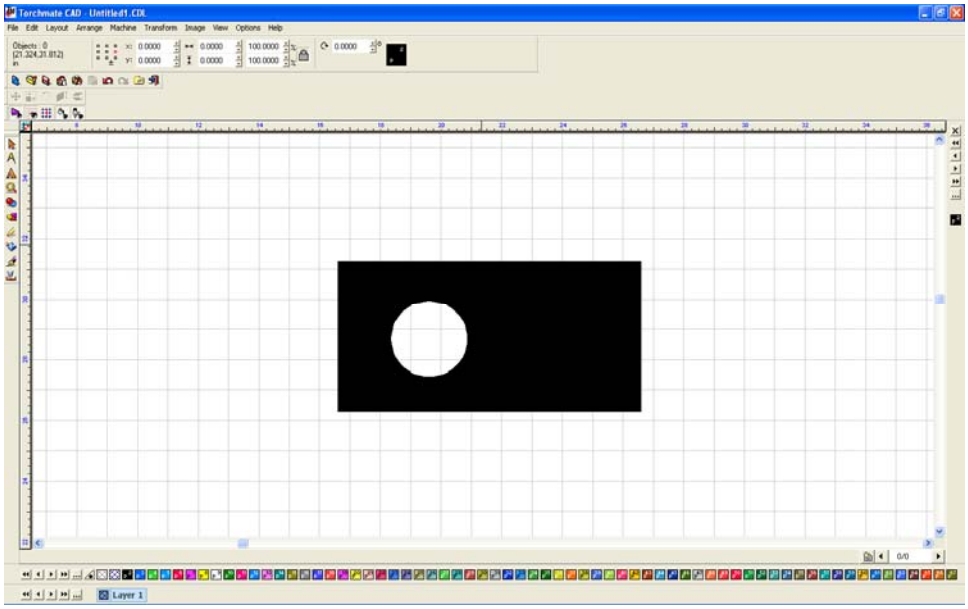
With both objects selected go to ‘Arrange’ → ‘Make Path’. This option tells the CAD that all objects selected are one part, allowing the software to determine if something is a hole or cutout.



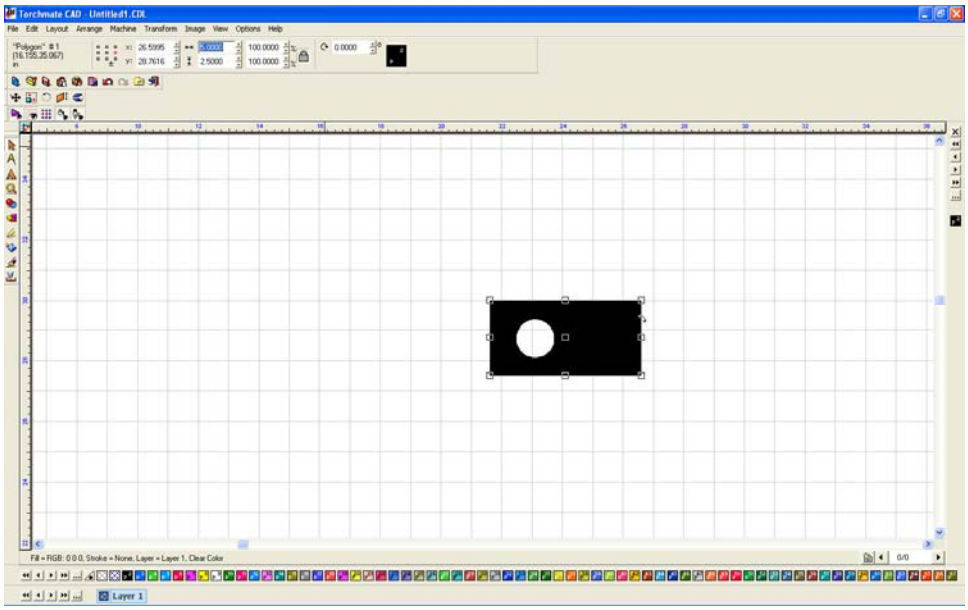
To verify that Torchmate CAD perceives the part as it is intended go to ‘View’ →’Show Fill’ this option displays the part as the computer sees it.



With ‘Show Fill’ on the part will fill in what is perceived as the part. Holes should be empty with the metal part filled in.



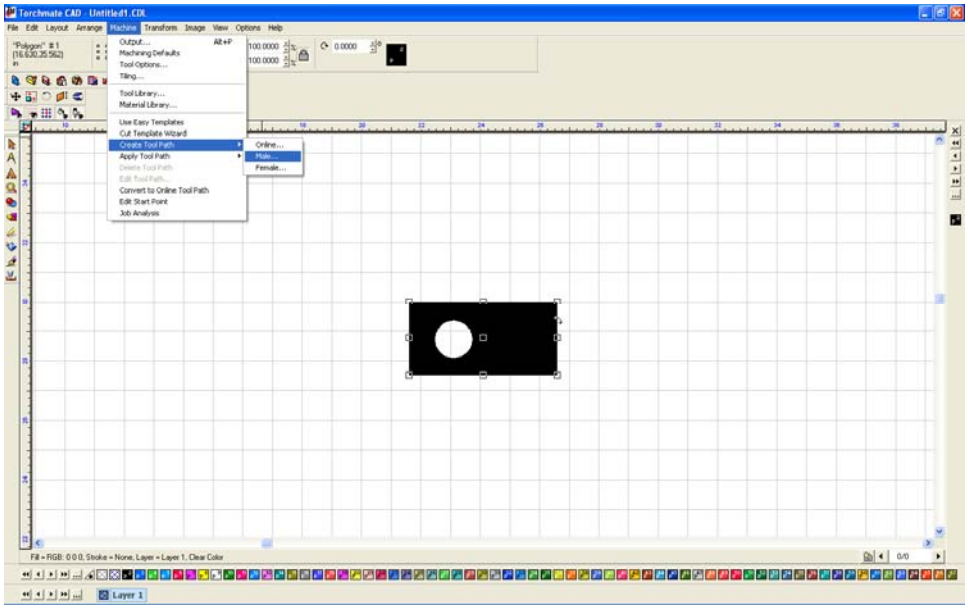
Once the part is finished it can be resized again if necessary, this resizing will adjust the hole's size along with the part's size.



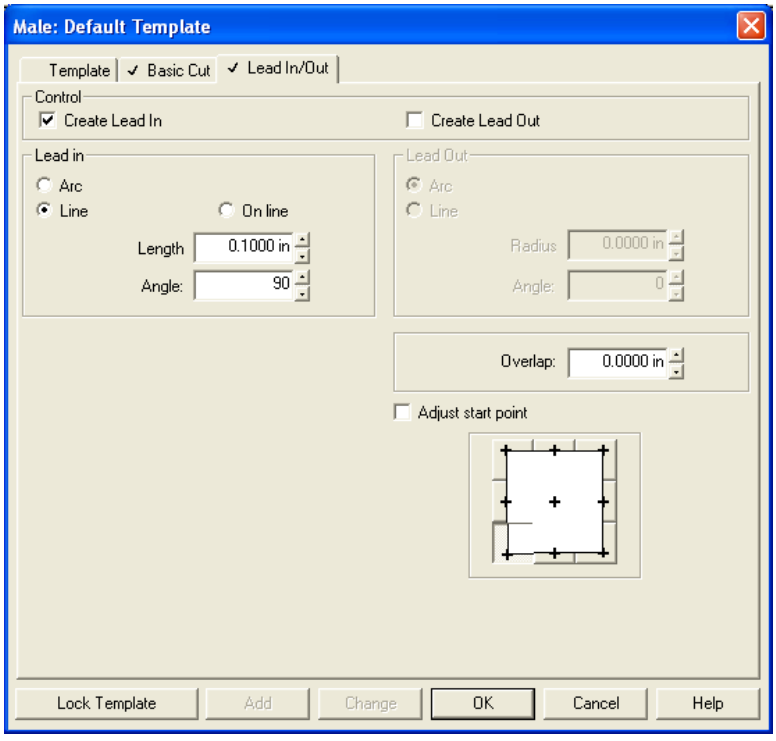
If the part is the correct size and 'Show Fill' appears correct the part is ready to have a tool path created.

Creating Tool Paths:

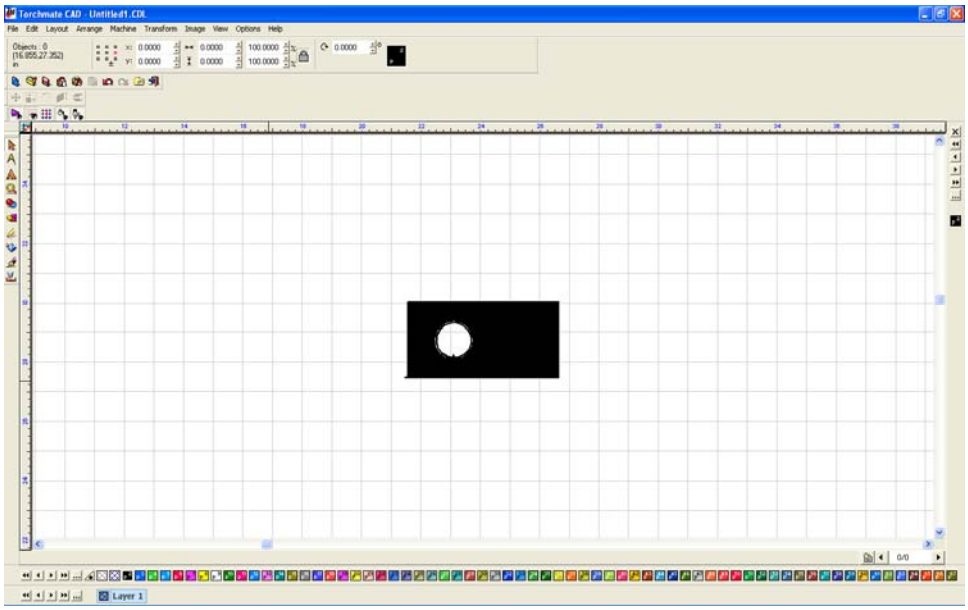
To create a tool path go to 'Machine' → 'Create Tool Path' → 'Male'.



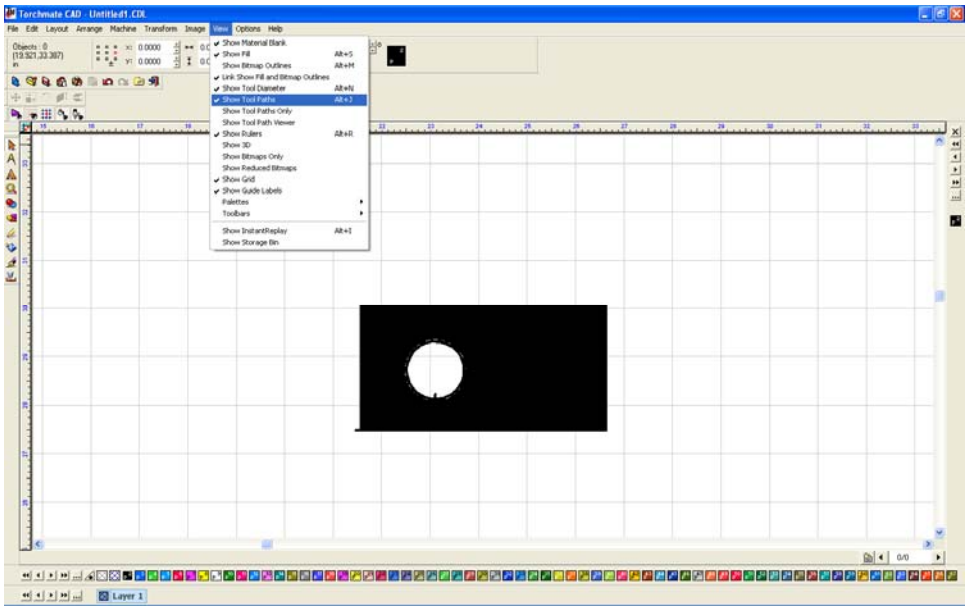
For this simple part the only changes to be made are under the 'Lead In/Out' tab. Create a lead in by specifying a length and angle. Then press 'OK' to create the tool path.



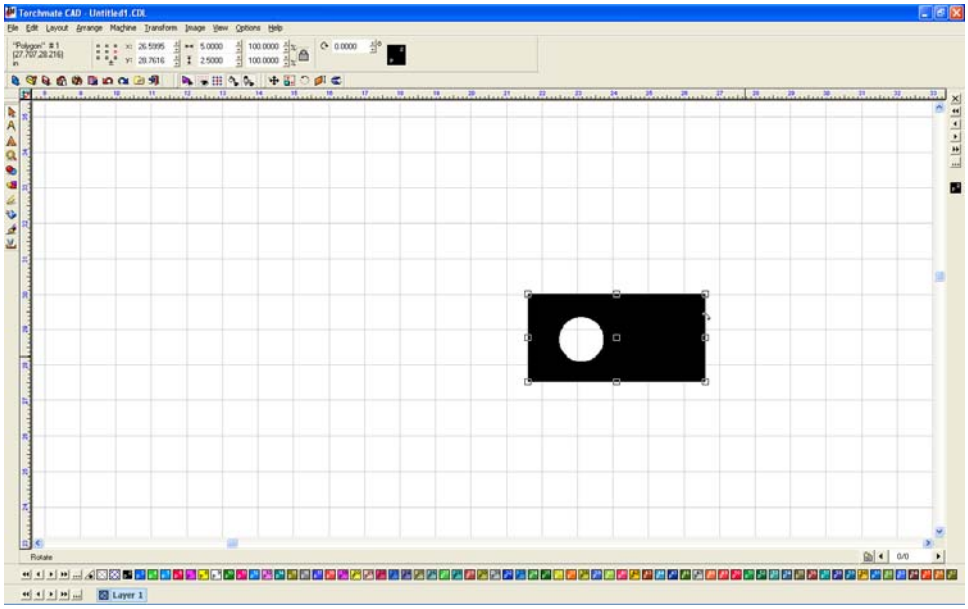
The screen now has two objects on it, the original part drawing and the tool path. The original drawing is no longer needed and needs to be removed.



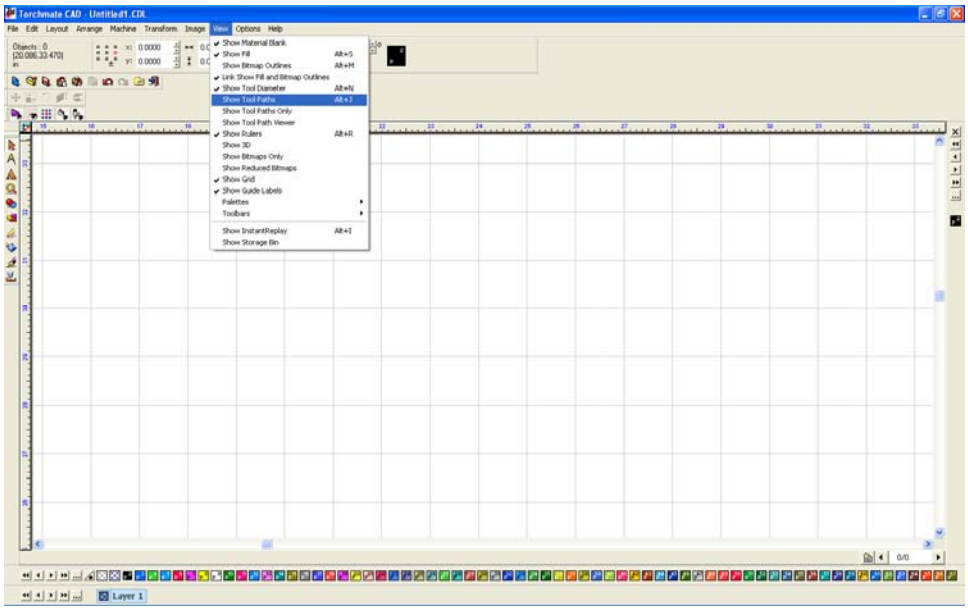
To eliminate the original drawing go to 'View'→ 'Show Tool Paths' and ensure that it is unchecked.



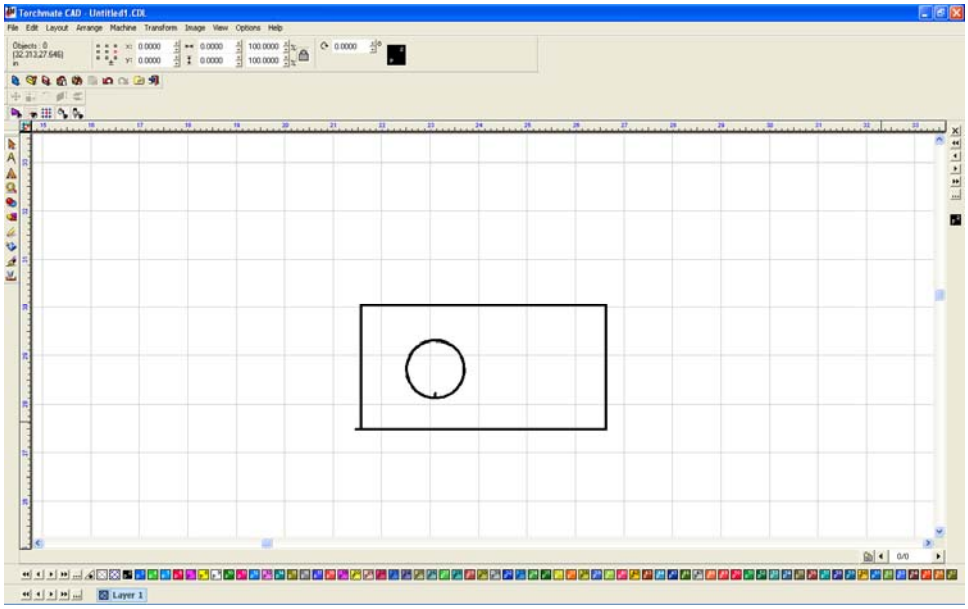
The only object visible now is the original object.



Select the object and press ‘Delete’ the screen will be blank now. Go back to ‘View’ → ‘Show Tool Paths’.

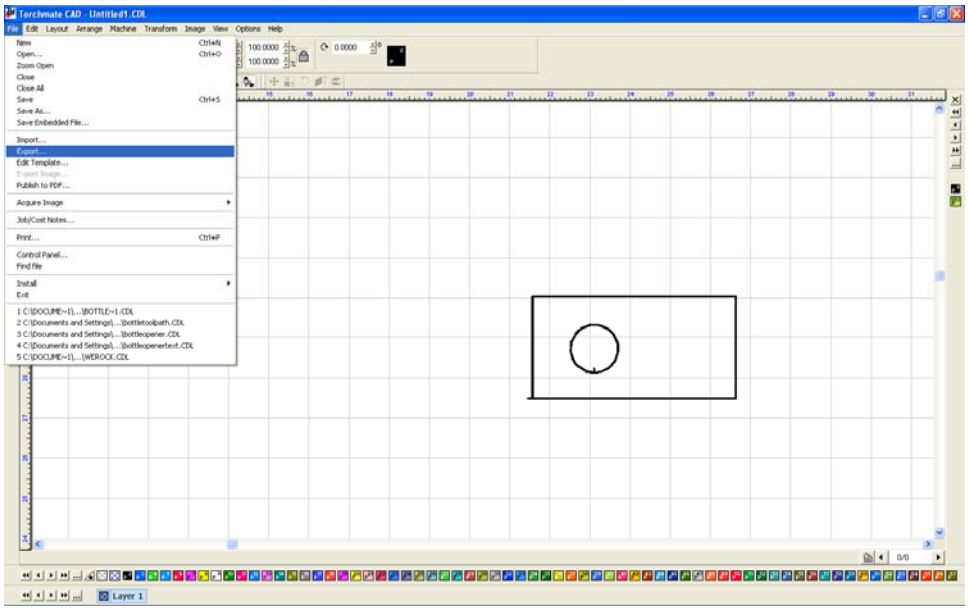


The only remaining object is the tool path, this part is now ready to export to be cut.

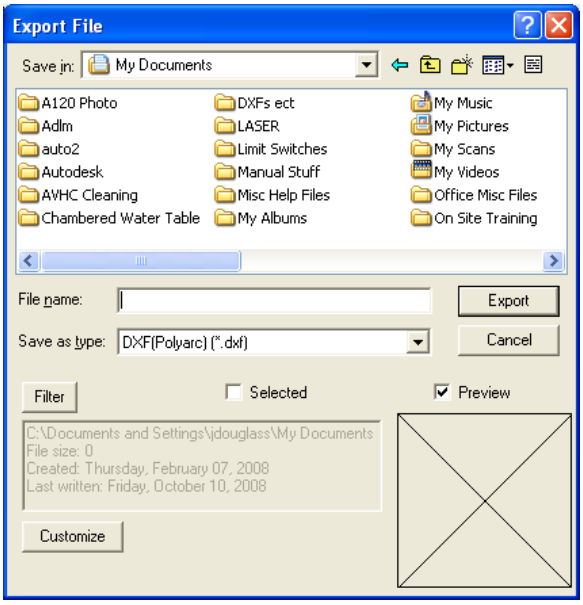


Exporting Files:

To be able to cut a part it needs to be exported as a 'DXF' format. To export go to 'File' → 'Export'.



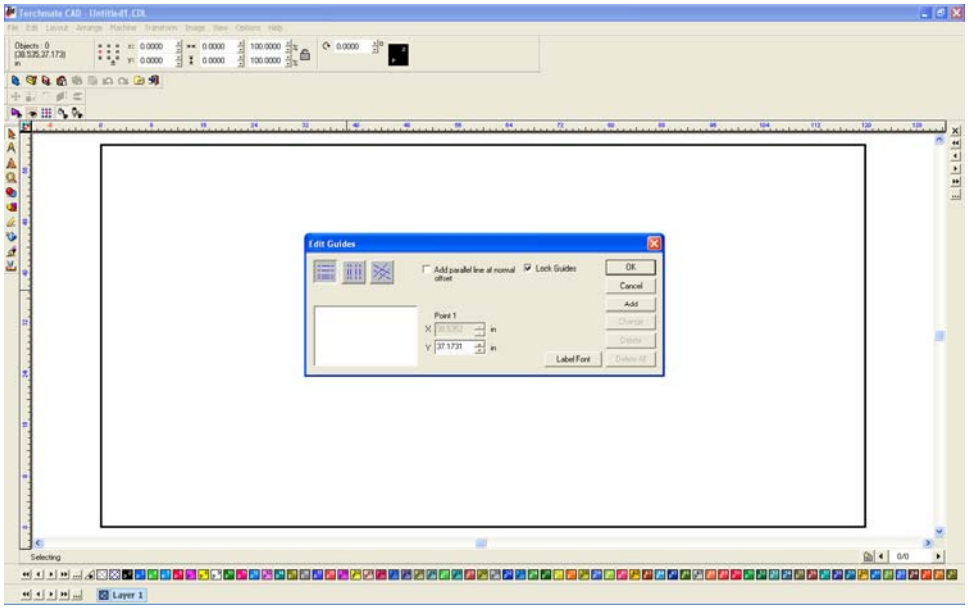
Ensure that the 'Save as type' is listed as 'DXF (Polyarc)'. Give the file a name and ensure that the location of where it is being saved is known. Select 'Export' and the file will now be able to be imported into the driver software to cut.



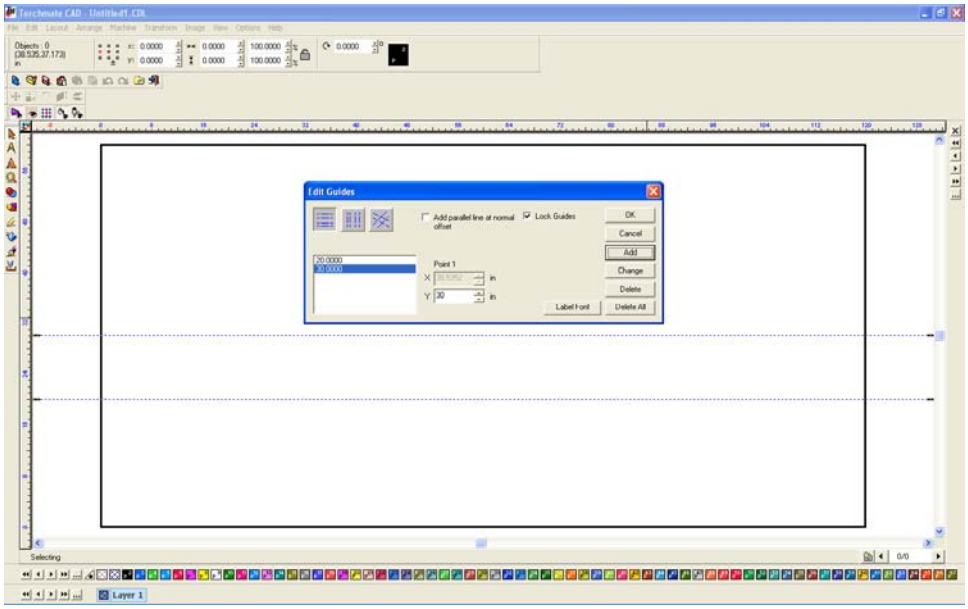
Using Guides and Alignment Tools:

Guide Tools:

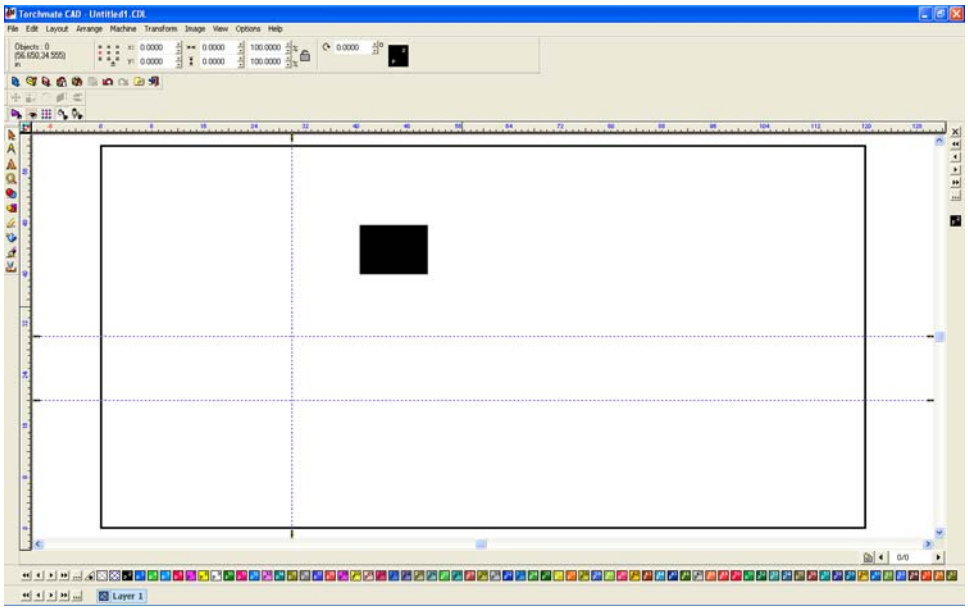
In Torchmate CAD guides can be used to align objects and nodes. To access the 'Edit Guides' menu have no object selected and right-click on empty space, the menu can also be accessed by going to 'Options' select 'Guides' and 'Edit Guides'.



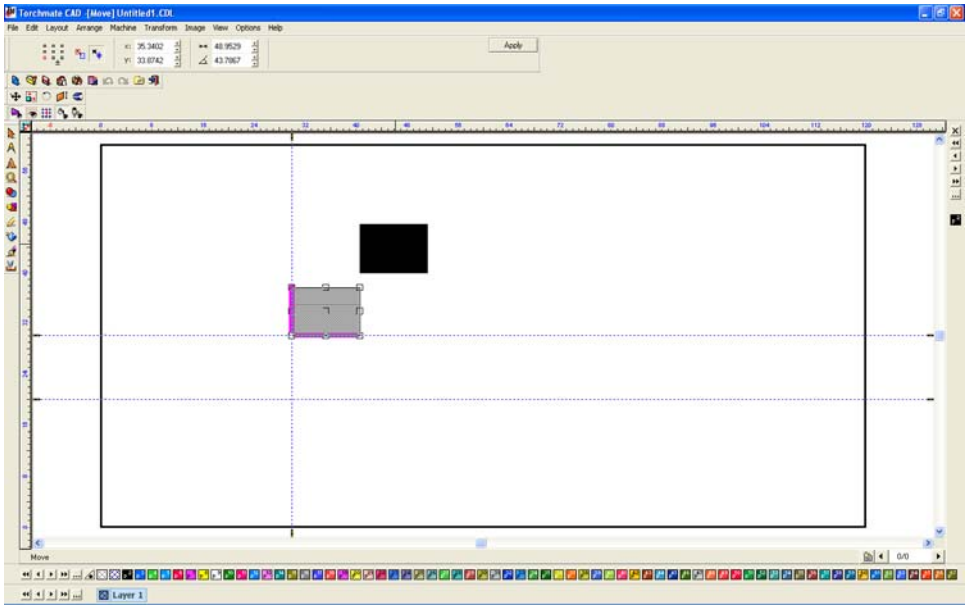
Guides use the coordinates related with the rulers. To add a line type in the coordinate the line will be at and select add. Horizontal, vertical, and diagonal lines can be added.



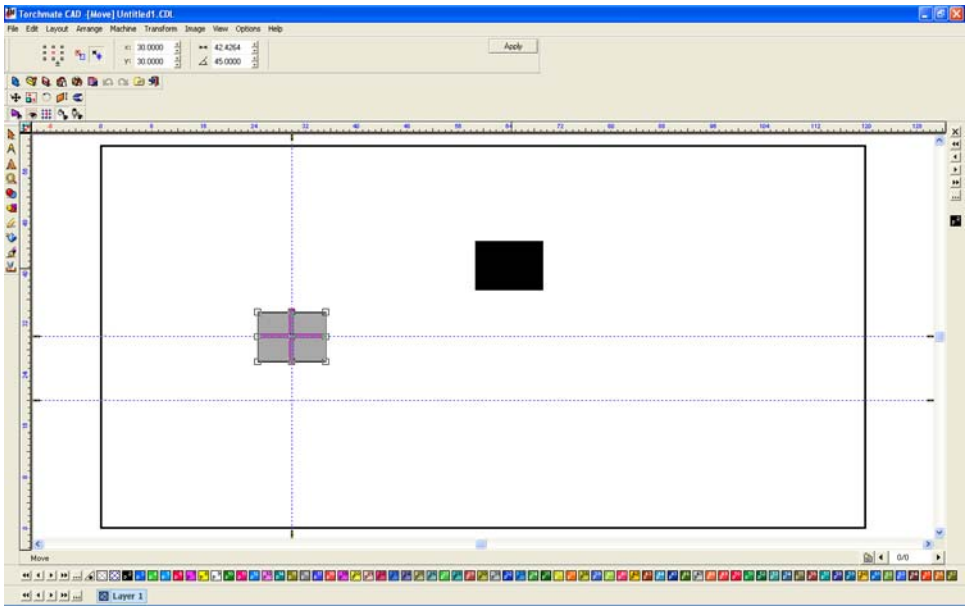
Once guides are created an object can be ‘snapped’ to the lines.



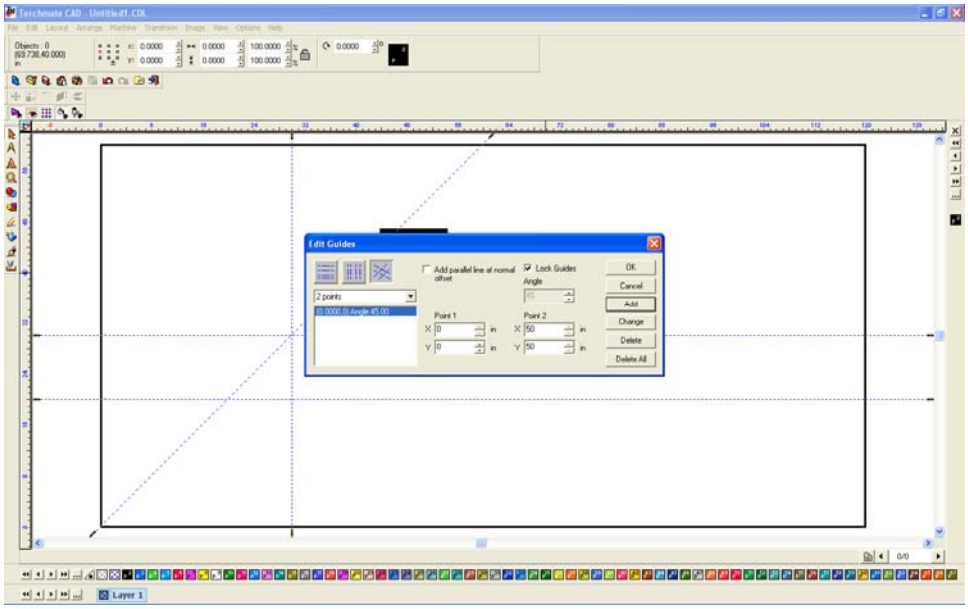
To snap an object to the guides move the object over to the guides, when the edge of an object turns purple the object is aligned to the guide.



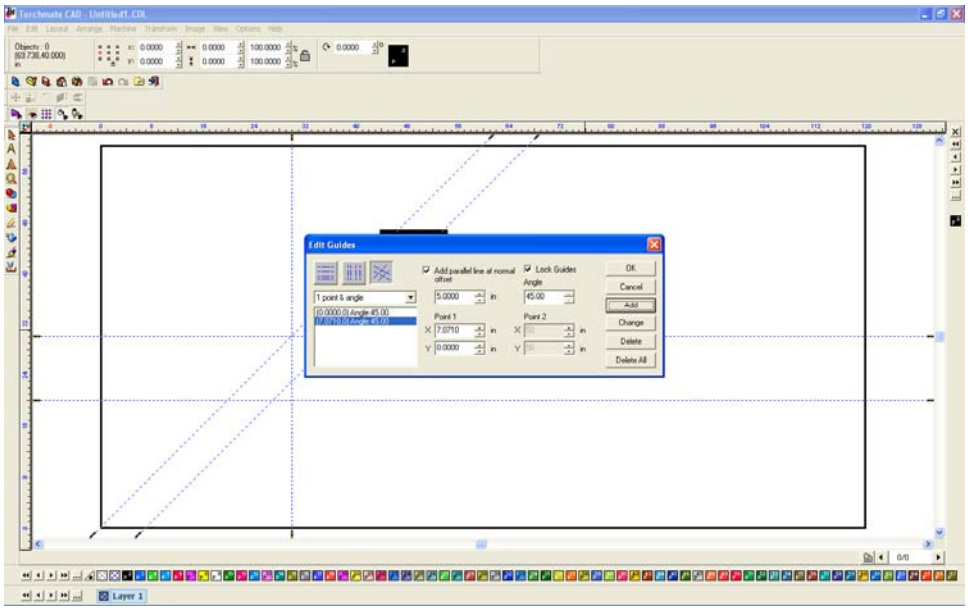
Objects can be snapped to any edge as well as to the center line of the object.



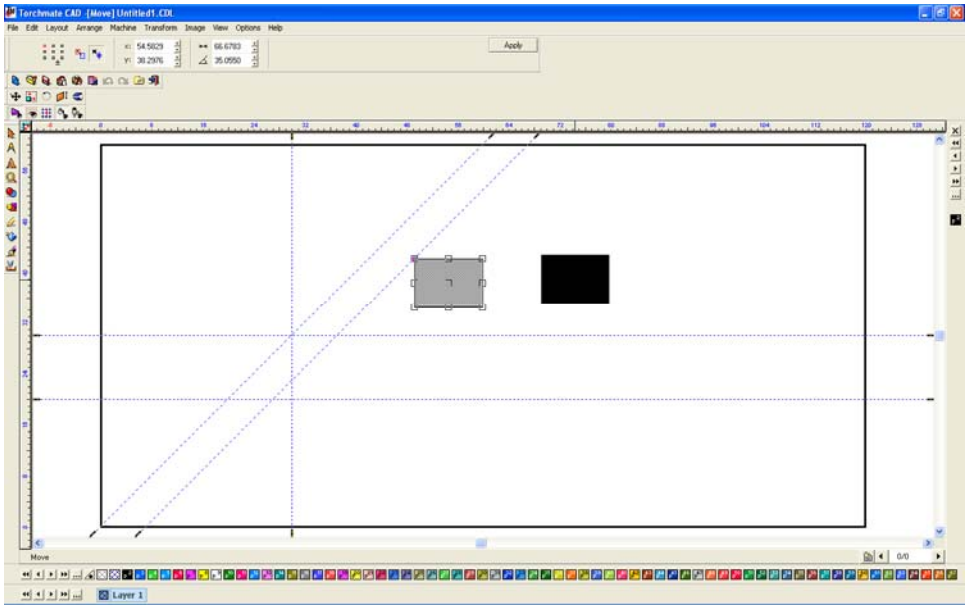
A diagonal line can be added two different ways, either by specifying a single point to pass through and an angle or by specifying two points that the line should pass through.



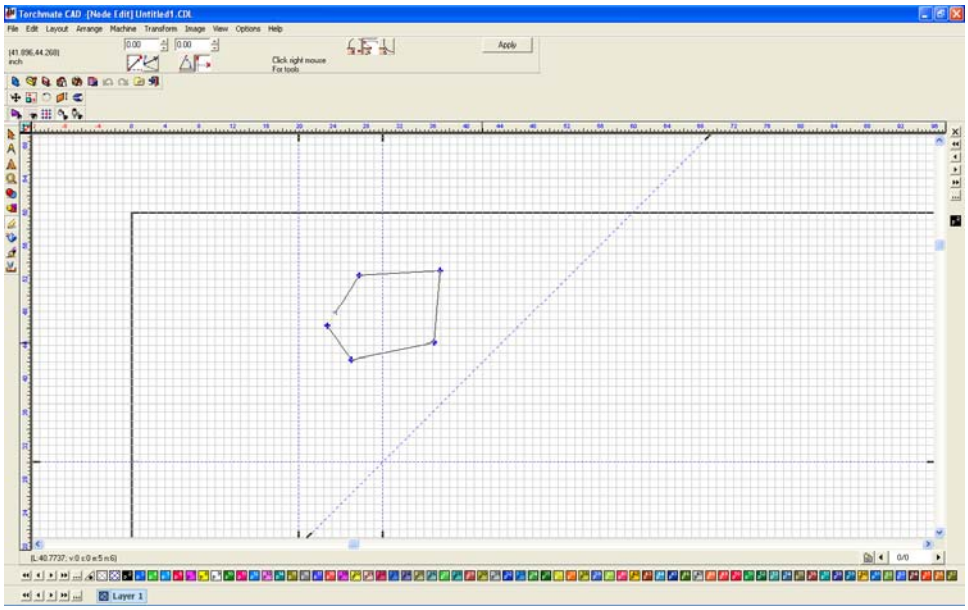
Any line can be added parallel to an existing line by selecting the ‘Add parallel line at normal offset’ check box. A new box will appear to enter how much of an offset should be used.



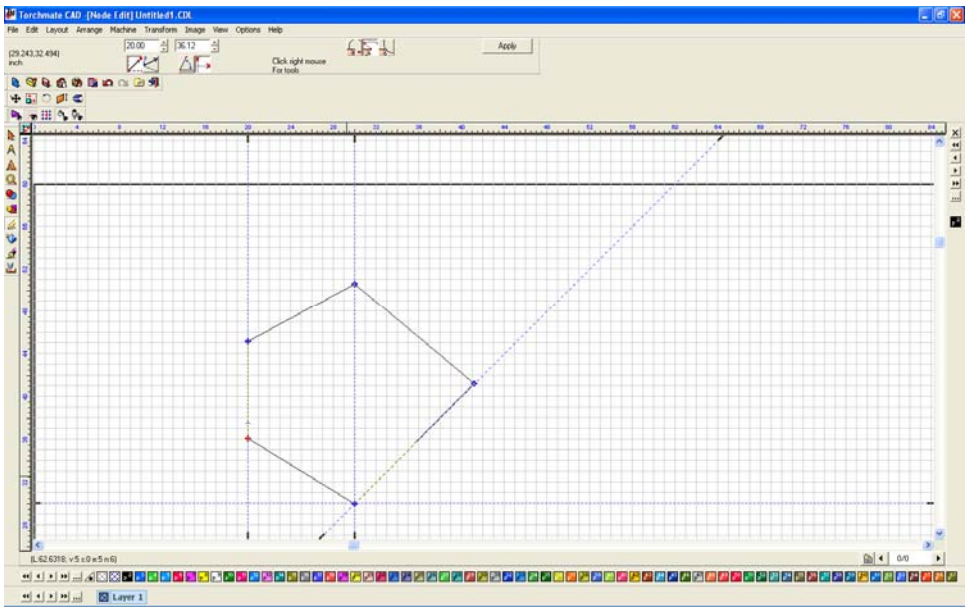
With diagonal lines an object can also be snapped to a corner, it will turn purple similar to an edge or the center.



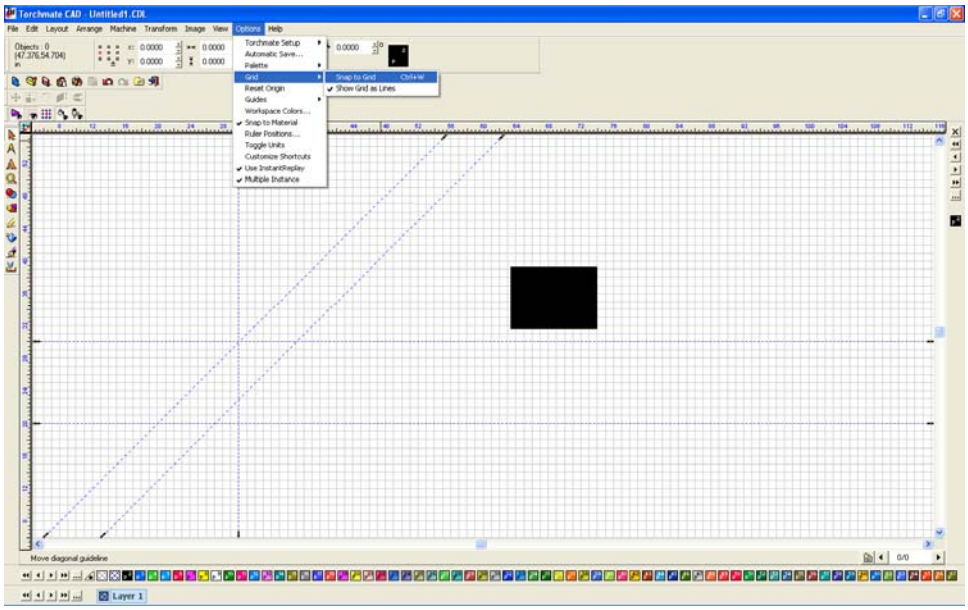
Guide lines work for nodes as well as objects.



Nodes will snap to the guide lines when they are moved near them, this allows for multiple lines to be aligned to each other.

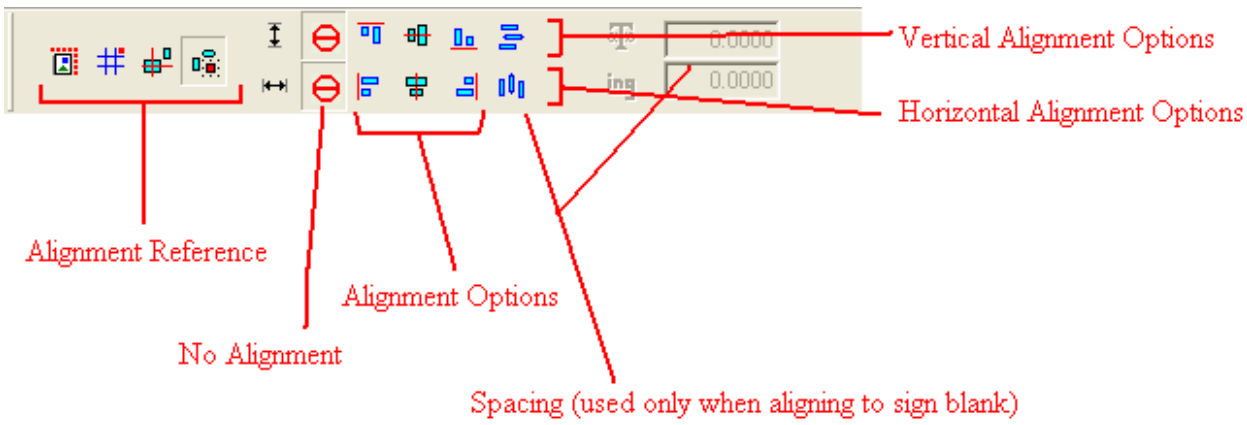


Another way of aligning objects is to go to ‘Options’ select ‘Grid’ and then ‘Snap to grid’ with this option enabled the object’s edge will attempt to align with the nearest grid line.



Alignment Tools:

If two objects are created that need to be aligned another method is the alignment tools. Go to 'Layout' select 'Arrange and Distribute' and then 'Alignment' the top tool bar will change to the alignment options. 'Alignment' opens the full options the other choices in the 'Arrange and Distribute' are quick links to the alignment options.



The alignment reference area specifies what the objects are being aligned to.

Align to Sign Blank – Aligns the selected objects to the material guide.

Align to Grid – Adjusts the position so that the objects selected align to the nearest grid line.

Align to Last Object – Aligns to the last object created of those selected.

Align to Selected – Aligns the objects based on what objects are selected.

For both the vertical and horizontal alignment there are four options.

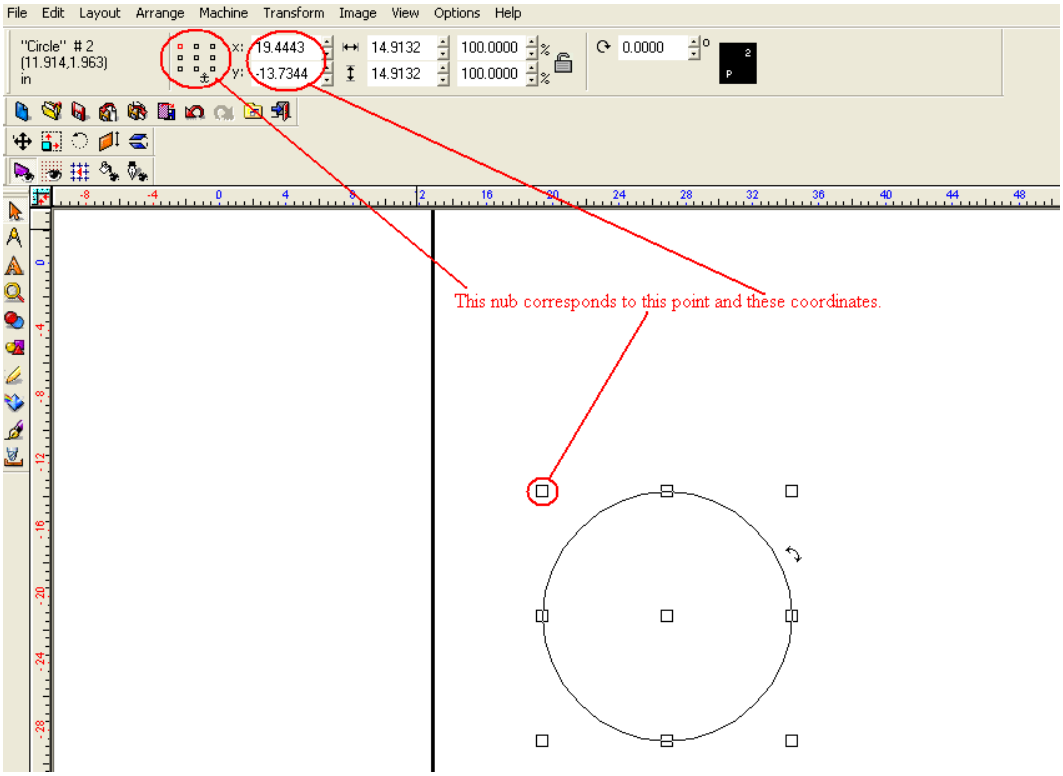
None – Does not change the alignment along this axis.

Top/Left, Bottom/Right – Aligns the edges of the object to the direction(s) specified.

Center – Aligns the objects centered to the reference area.

Spacing is used when aligning to the sign blank. This will space the objects equally across the material size. The values that are entered in the boxes adjust how far from the edge of the material the objects will be.

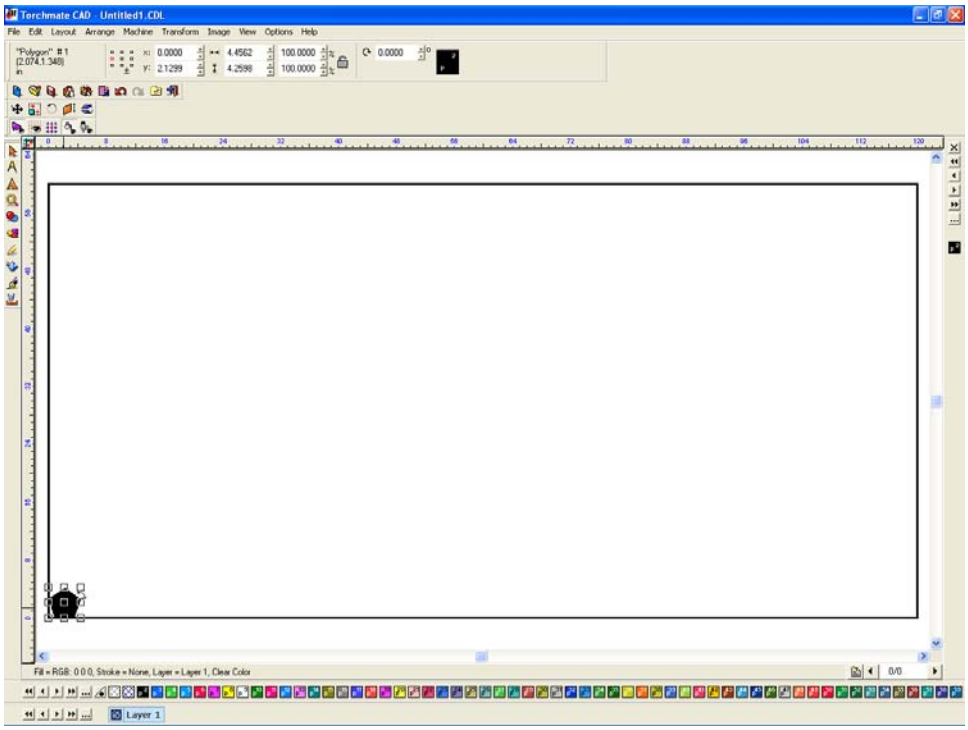
The last method of aligning objects is to use the ‘nub’ tabs on the top tool bar to reference coordinates. When an object is selected it will show nine black nubs next to the coordinate and measurement boxes. The nub that is red is selected; the coordinates displayed will correspond with the nub on the object.



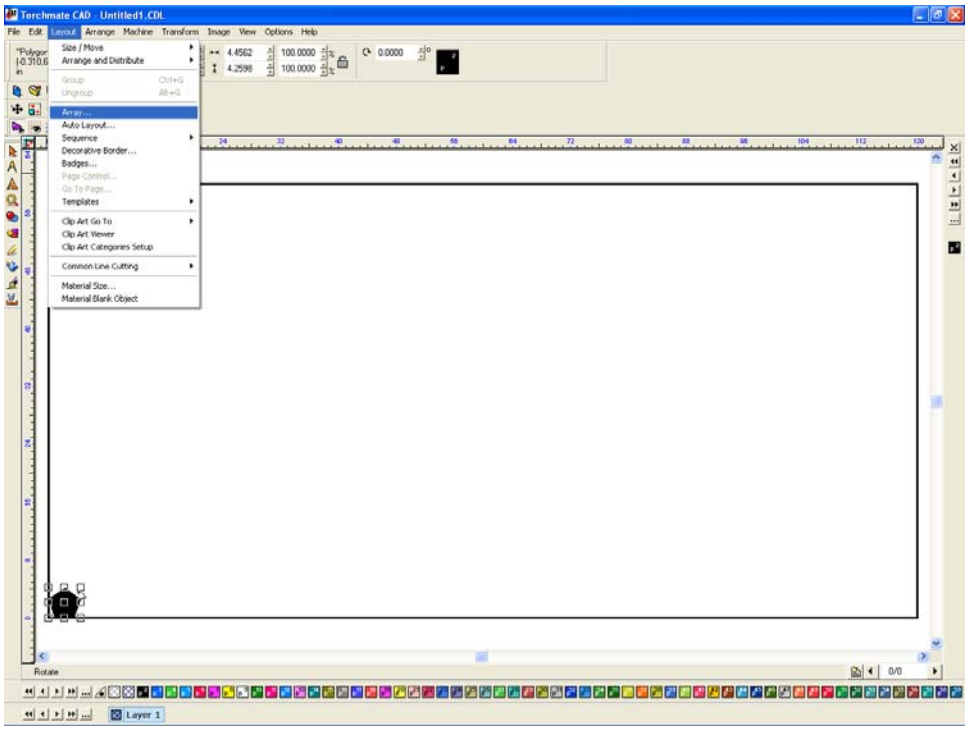
Using the alignment nubs two points on two objects can be aligned to the same coordinates simply by changing the coordinates in the boxes.

Using Array and Badges Features:

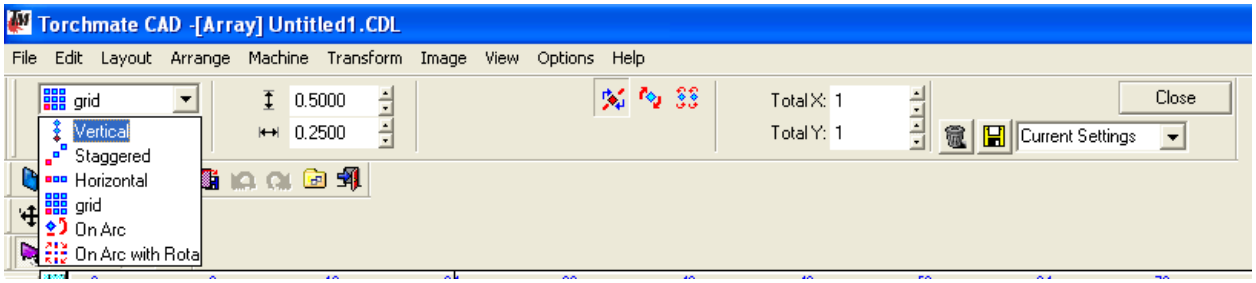
For applications that require creating multiple copies of the same part Torchmate CAD has two main tools, ‘Array’ and ‘Badges’, both are accessed through the layout menu.



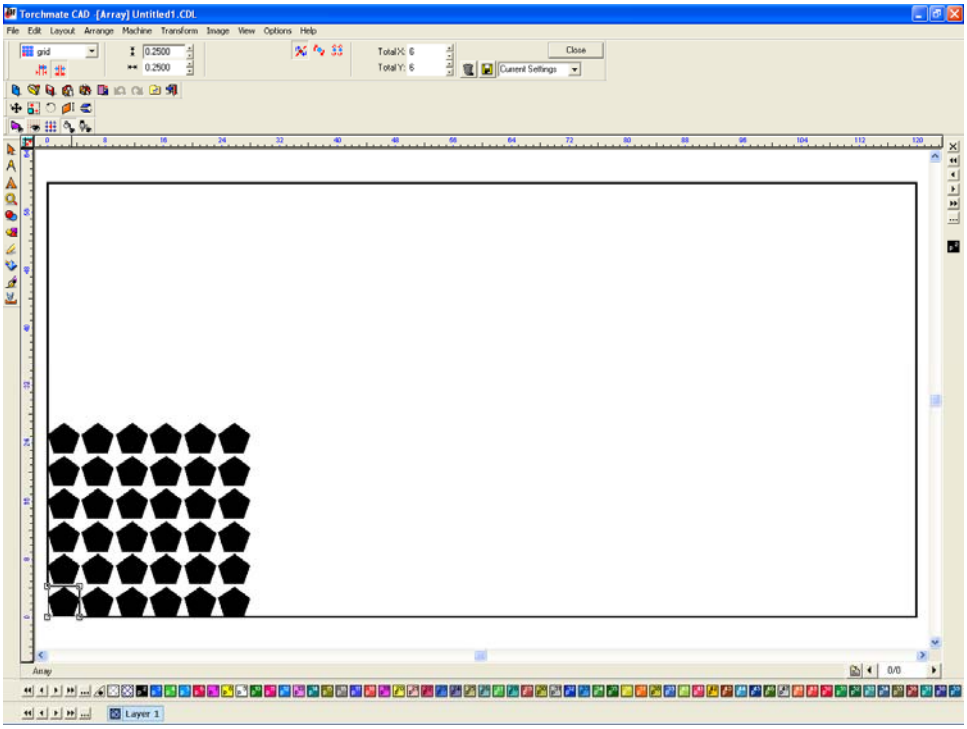
Array begins creating duplicates based on where the part currently is.



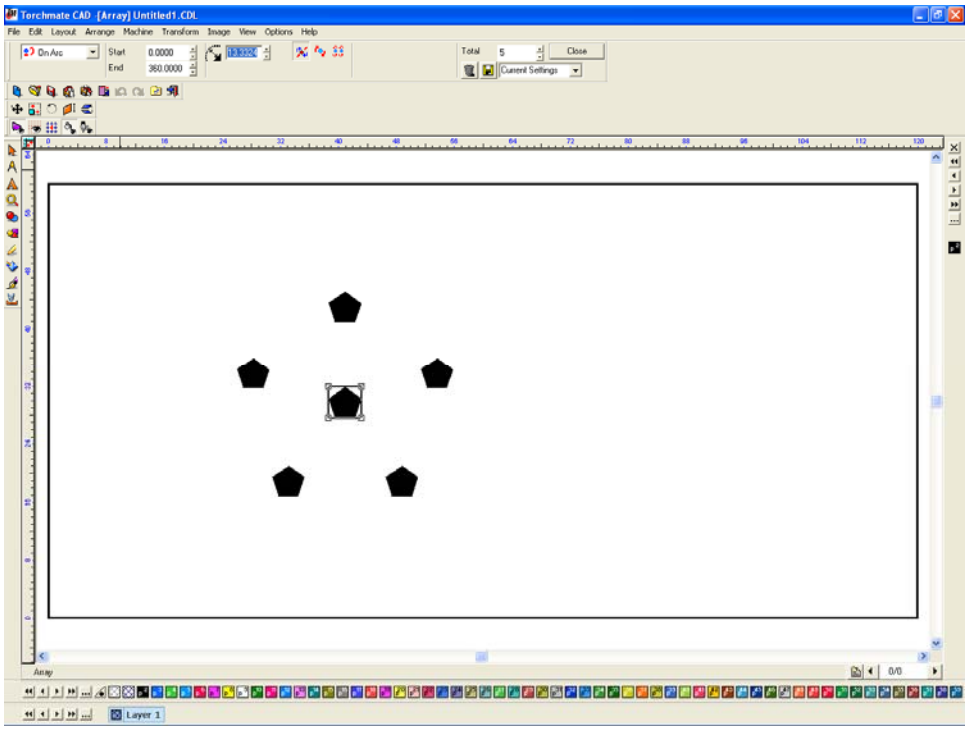
The array toolbar uses a pull down menu to select which type of array the CAD will create.



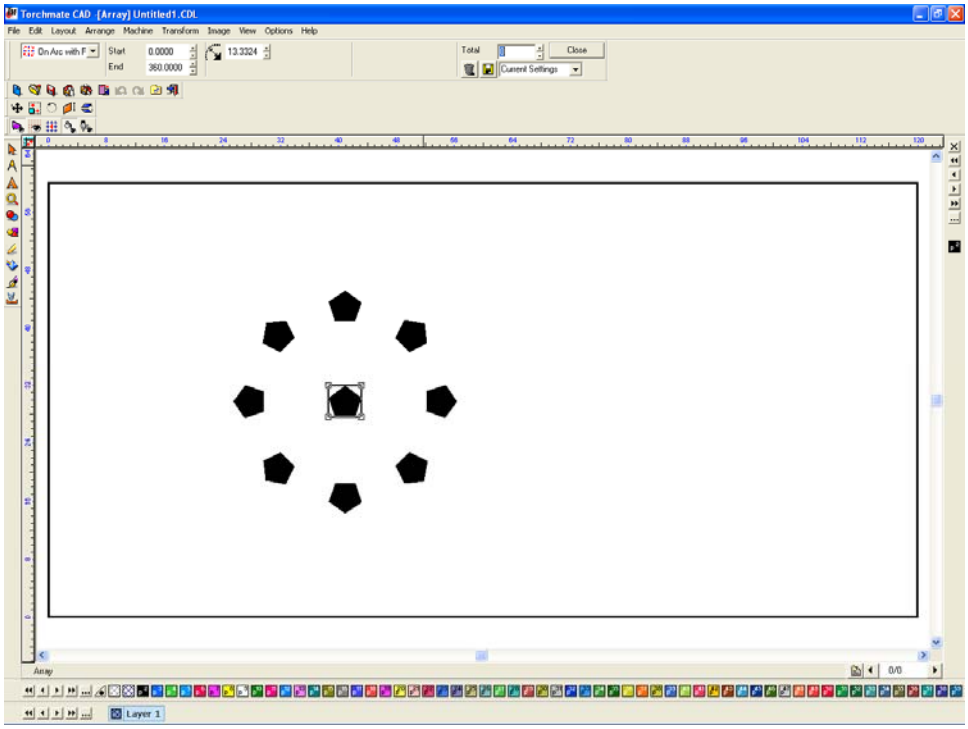
Grid, vertical, staggered, or horizontal all work on the same principle. The tool options allow for spacing along each axis as well as how many copies are desired.



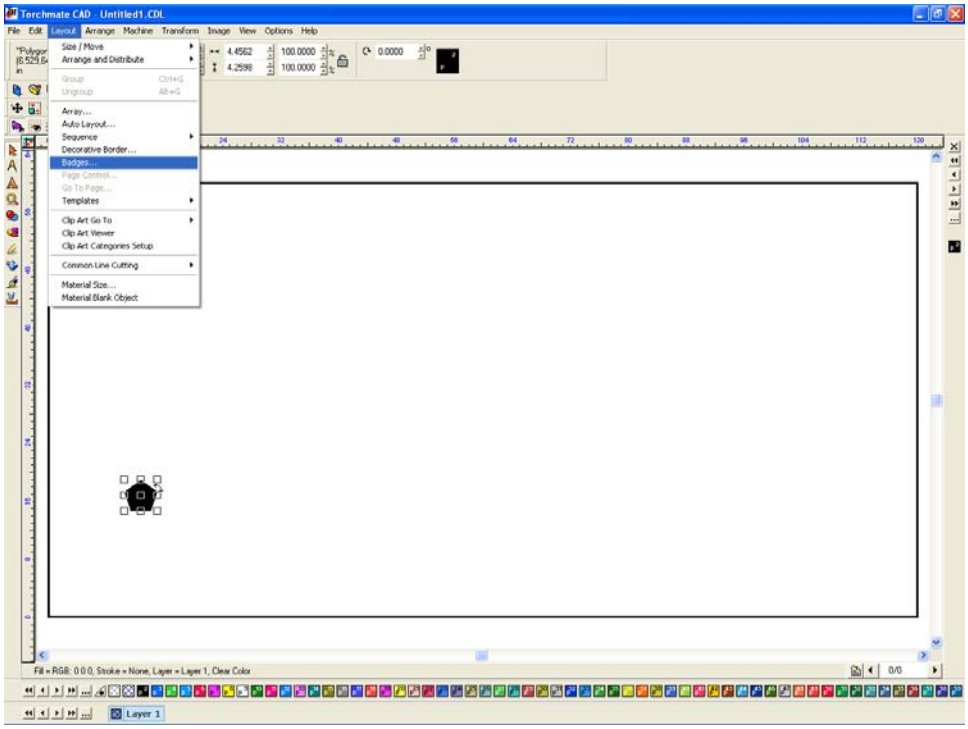
‘On Arc’ arrays specify how many copies and a distance from the center, the parts are then arranged equally around the original part equally spaced.



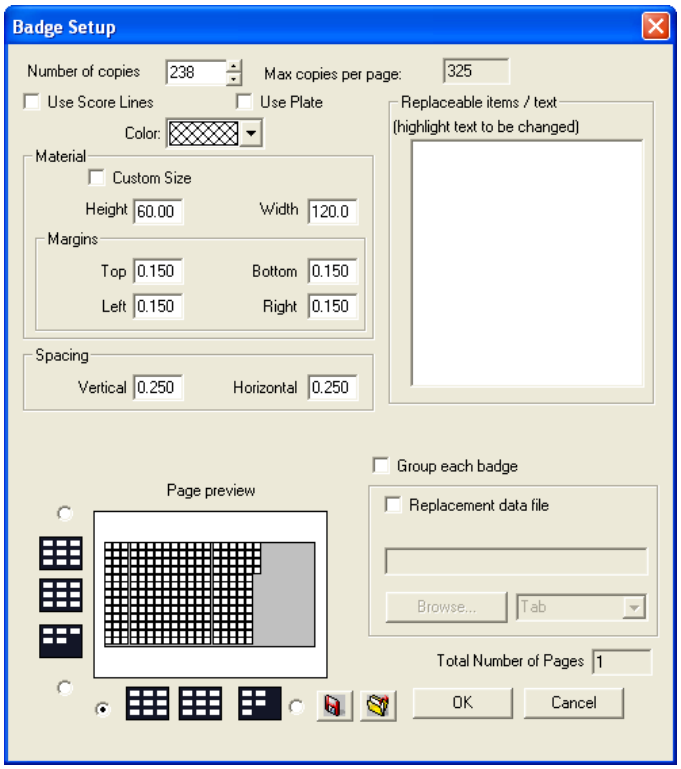
‘On Arc with Rotation’ creates the duplicates around the original equally spaced as the ‘On Arc’ does, but it will also ensure that the part’s base is facing toward the center.



Badges is similar to array but designed specifically to fill a sheet.



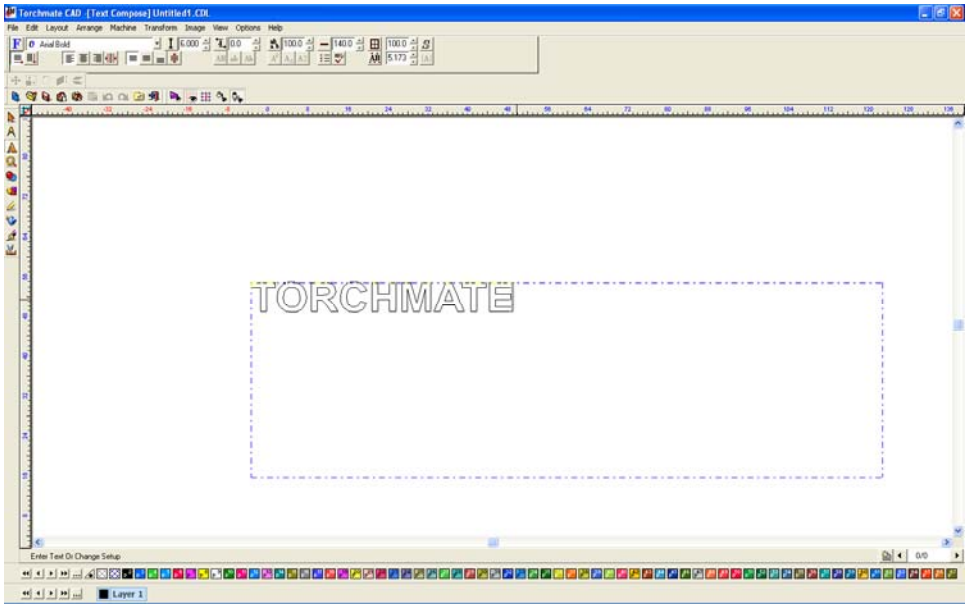
The 'Badge Setup' window allows the parameters to be set. On the upper right of the window the number of copies that can fit on the sheet with the current settings.



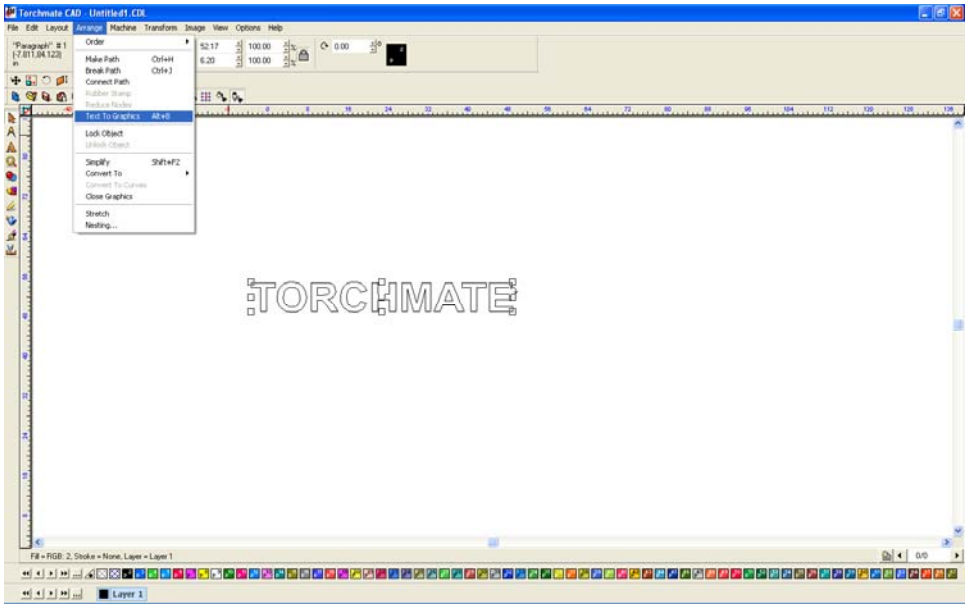
Editing Text:

Torchmate CAD allows various methods of editing text that can be used in conjunction with other tools to create fabulous results.

To start editing text select the text tool from the left toolbar. Select where the text needs to go and click, then type in the desired text.

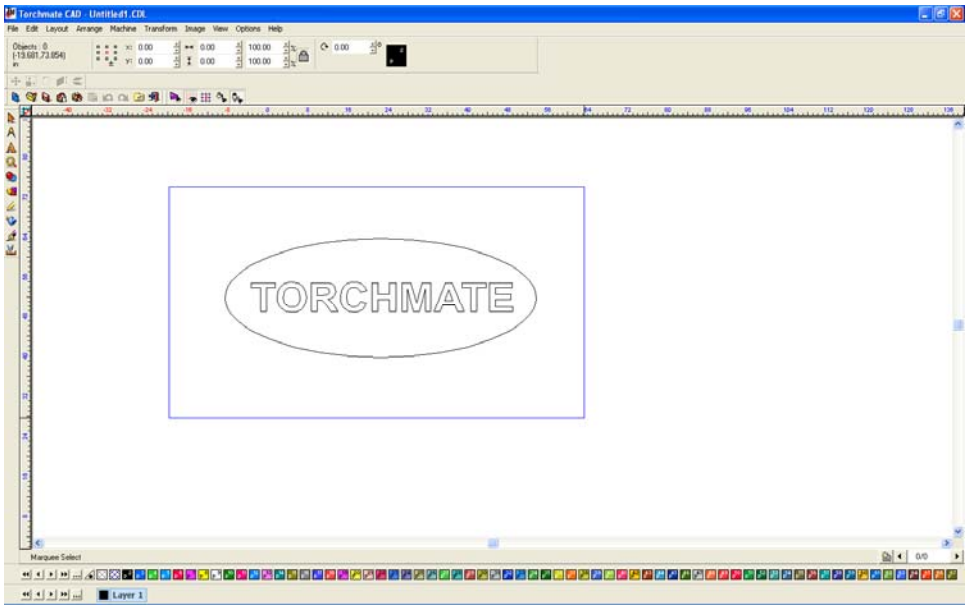


Once satisfied with the text in order to edit the text further it must be converted to a graphic. With the text selected go to 'Arrange' → 'Text to Graphics'. The letters are now individually selectable and can be manipulated as any other drawing. Examples below illustrate some basics of what can be done.

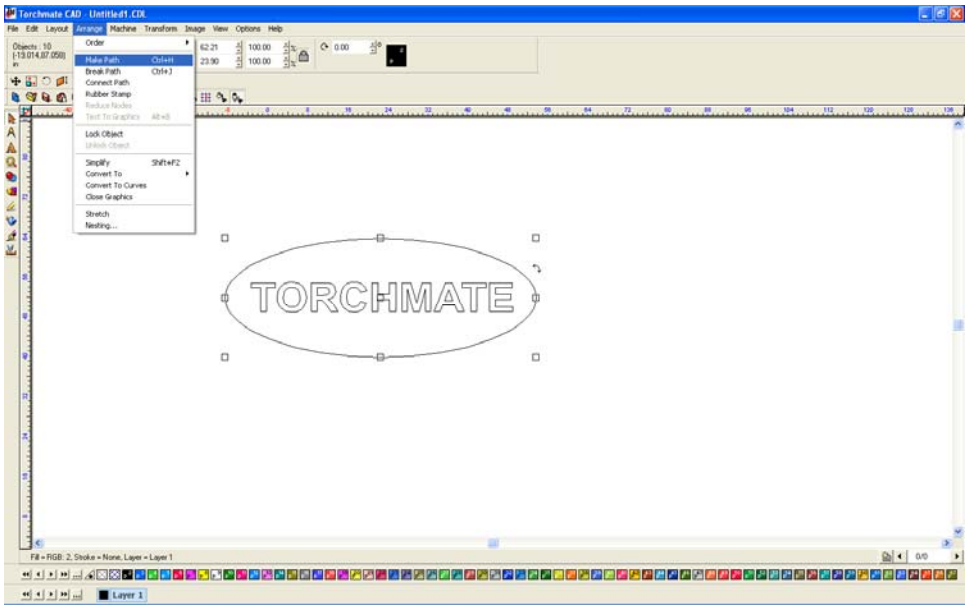


Cutting Text Out of an Existing Shape:

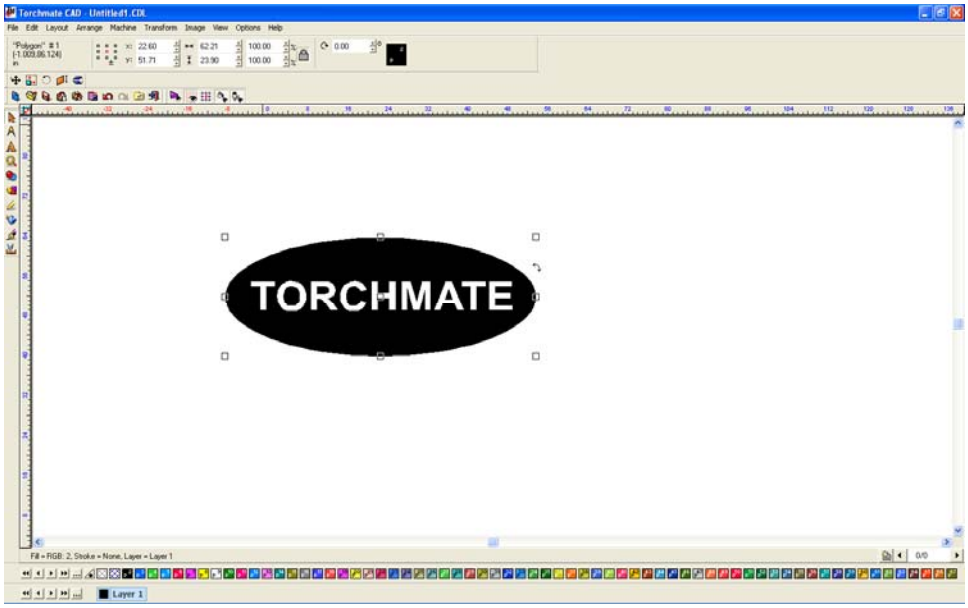
Once the text is created and converted to a graphic position it on the object being cut out of.



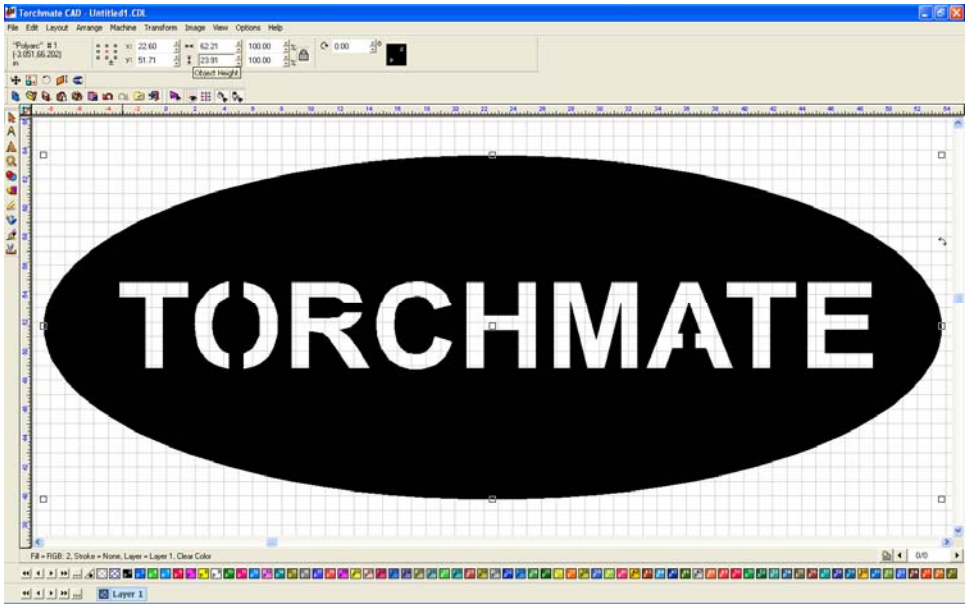
Once positioned go to ‘Arrange’ → ‘Make Path’. What this function does is tell the CAD any overlapping part is a cut out of the main part.



Once a part is done it is always a good idea to go to ‘View’ → ‘Show Fill’. Show fill displays the part as the computer sees it, white space indicating scrap areas. When working with text letters with cutouts like ‘O’, ‘B’, ‘A’, etc. the cutout will fall out if it is not bridged.

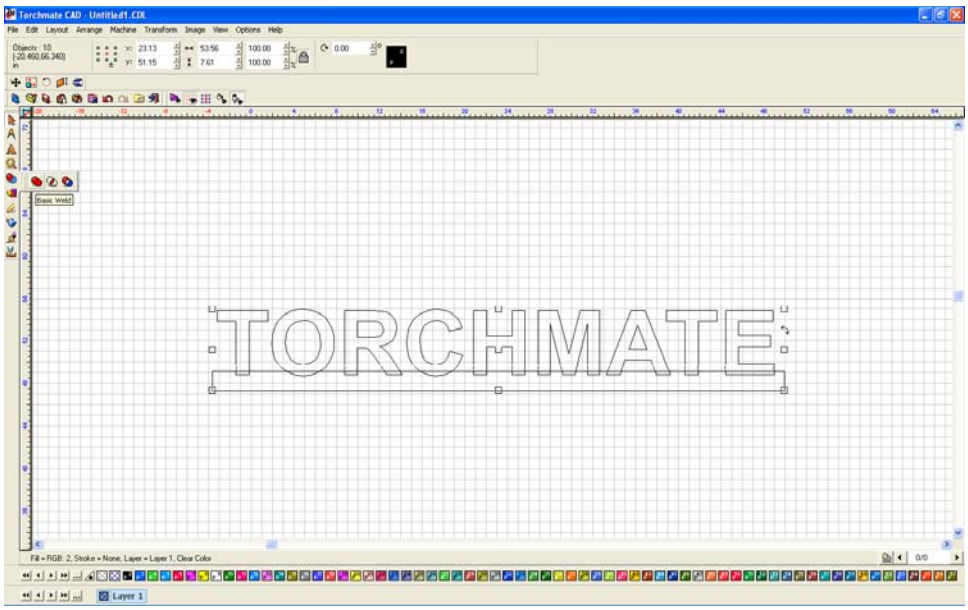


To bridge a letter simply draw a small rectangle over the area wanted to be bridged and complete a ‘Basic Weld’ from the weld menu on the left hand toolbar.

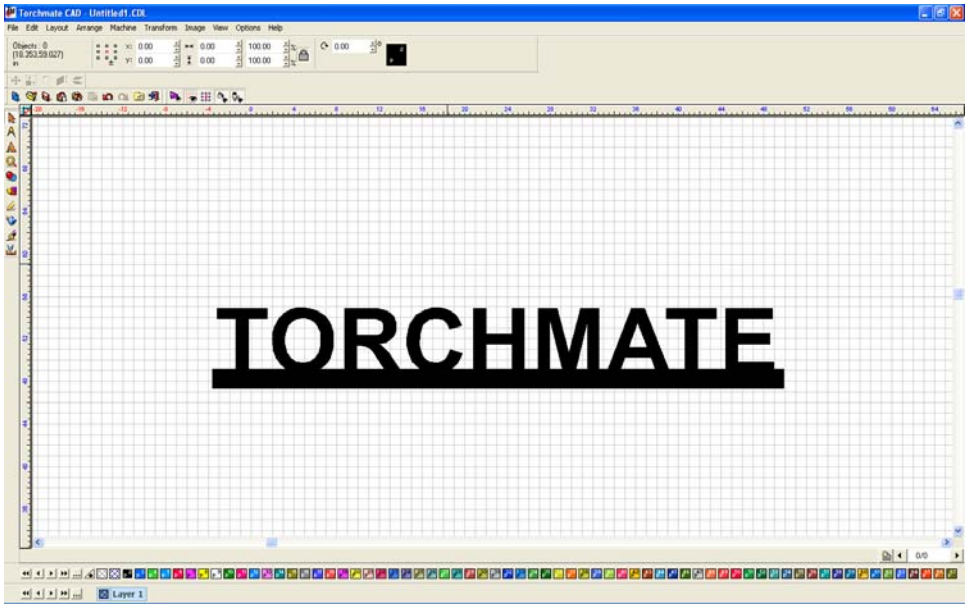


Welding Text to Objects:

Text can be attached to simple or complex objects. Once the text is converted to graphics be sure that the object slightly overlaps the text. Select all objects and text being joined and go to the weld tools menu. A 'Basic Weld' will merge all overlapping objects and text together.

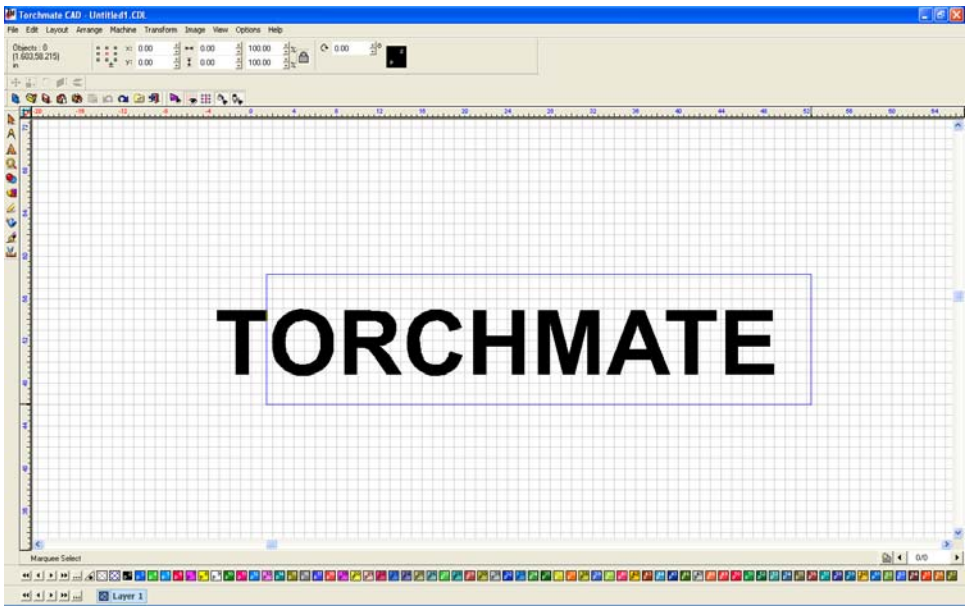


Be sure to turn on show fill once all editing is done to ensure that the CAD software is interpreting the drawing as it should be cut.

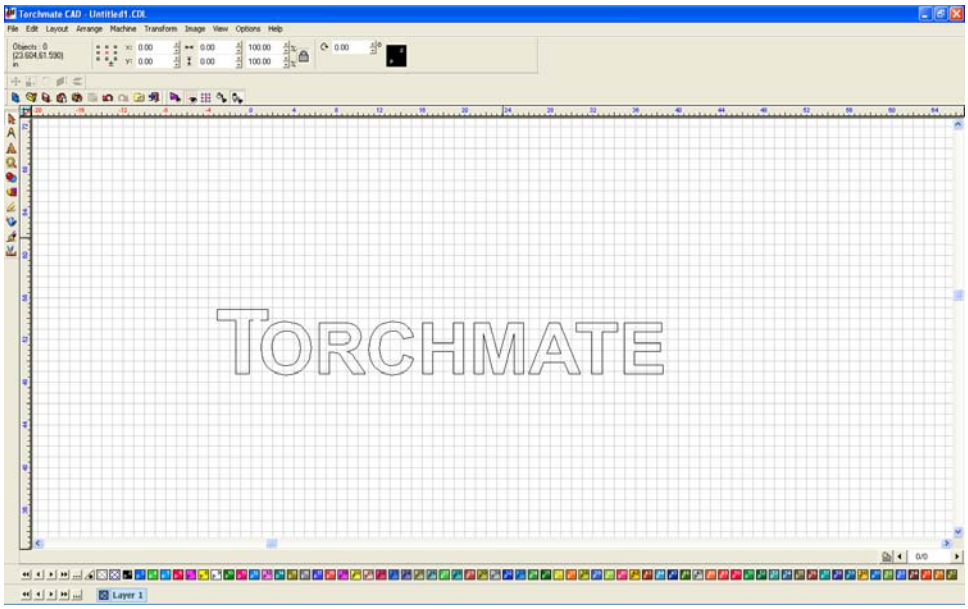


Resizing Text:

Text made in Torchmate CAD can be resized after it is converted to graphics. Simply select the letters needing to be resized. The letters can be resized like any other object, (using the corner tabs when selected or adjusting the scale from the top menu).

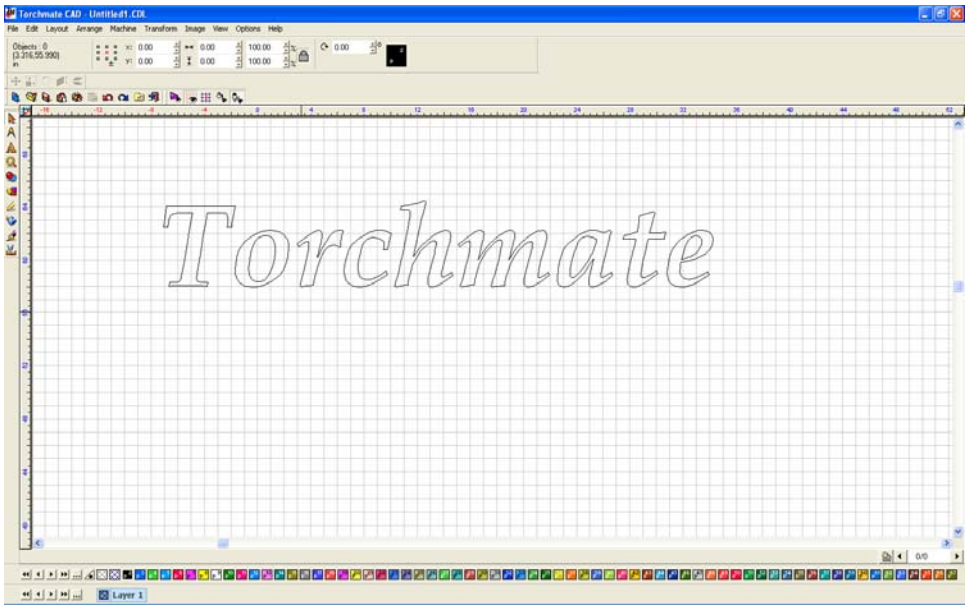


Once the letters are resized they can be manipulated any way normal letters can.

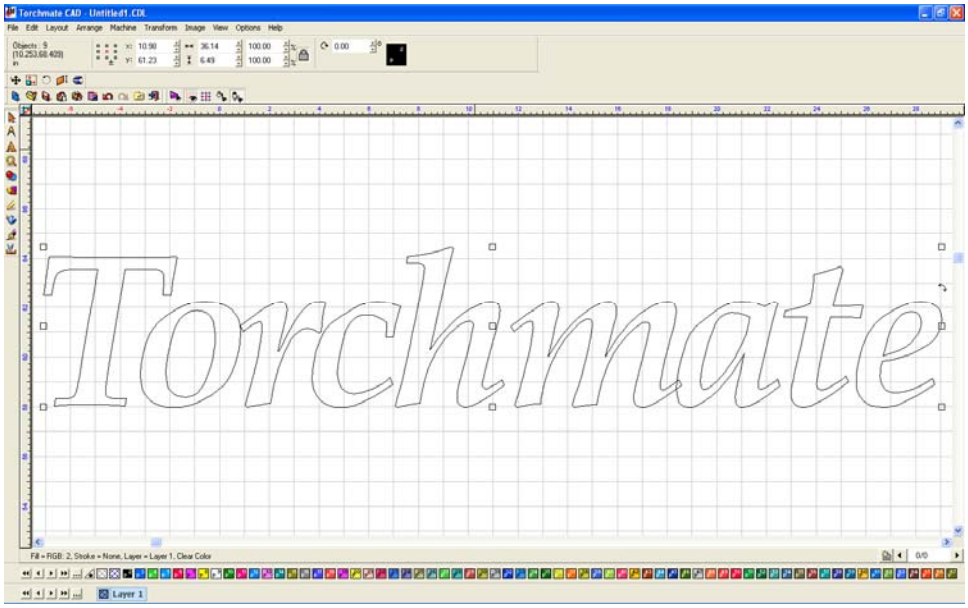


Joining Script Text:

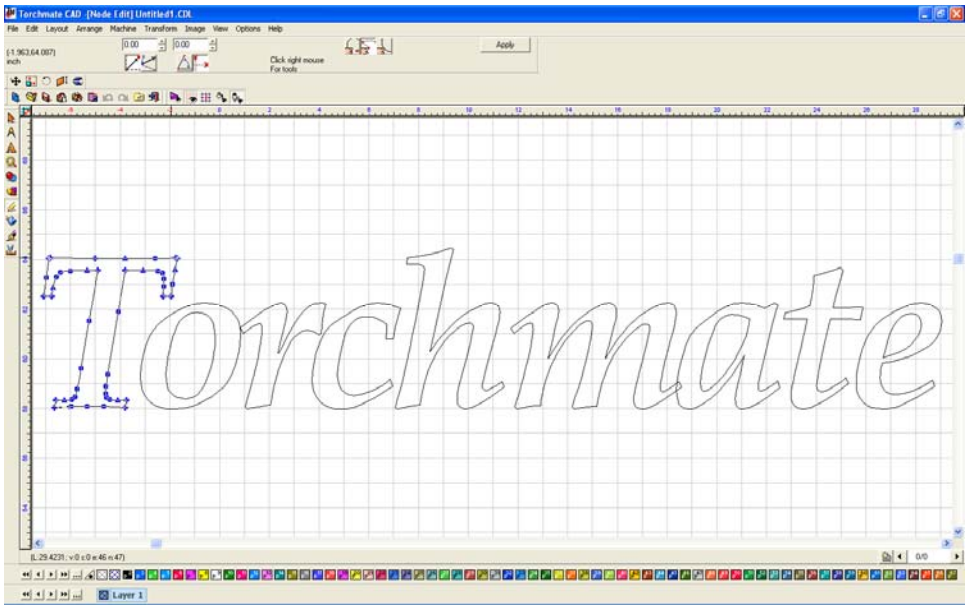
Node editing can be used to join script style text together to make a single cut for an entire word.



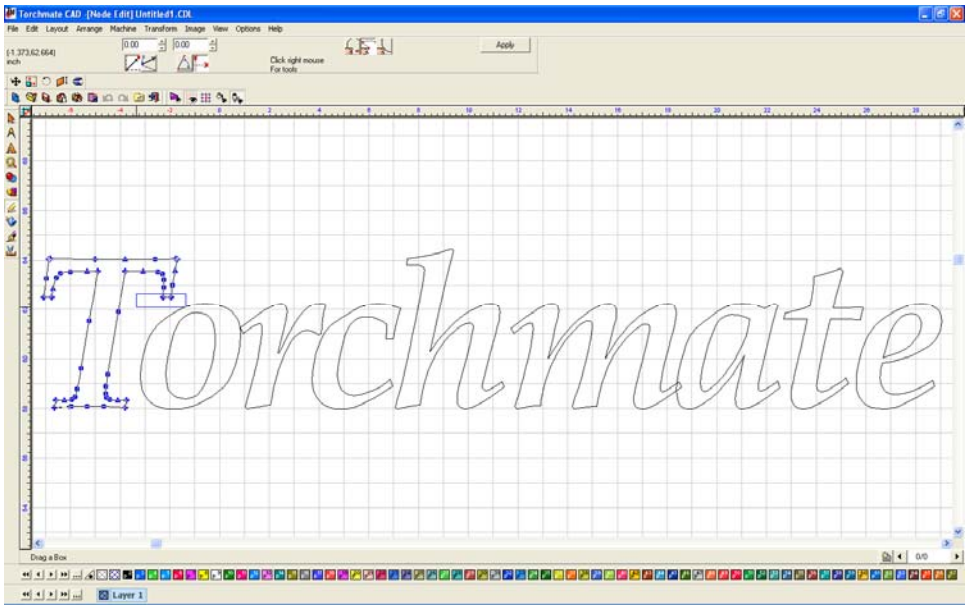
To start convert the text to graphics, then move each letter close together so that they can be joined.



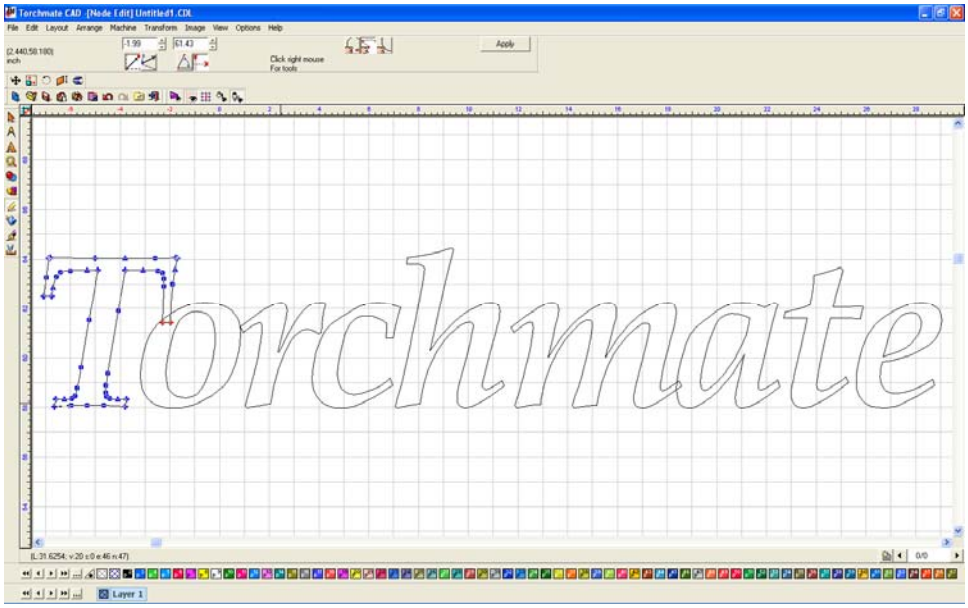
Double clicking on a letter will open node editing. With node editing the letter can be manipulated to any shape that is needed.



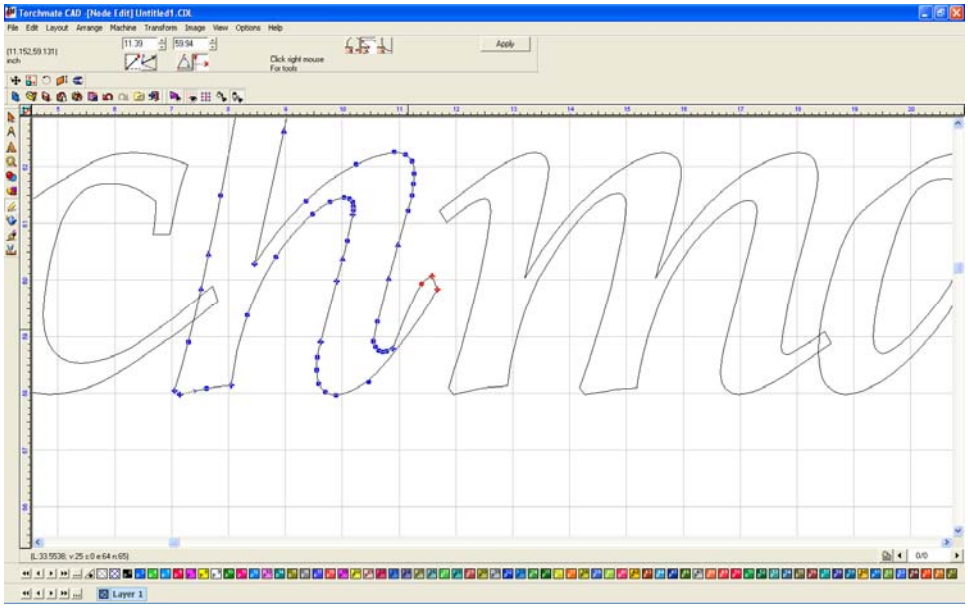
Unselected nodes are blue in color while a selected node will be red. To select a node click on it, or drag a selection box around a group of nodes. With a node or nodes selected they can be moved around.



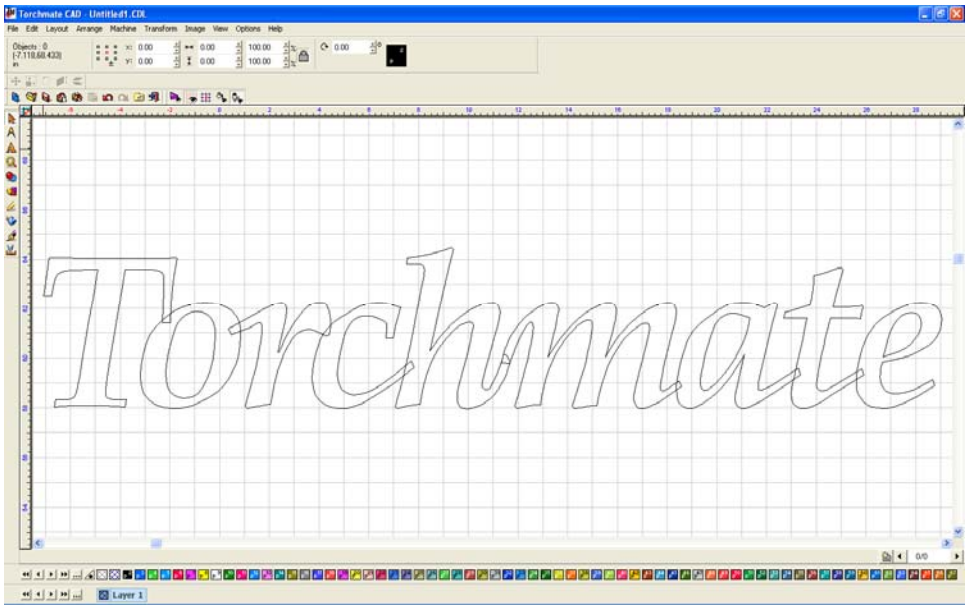
To join a script text together the goal is to overlap all letters and then weld the individual letters together.



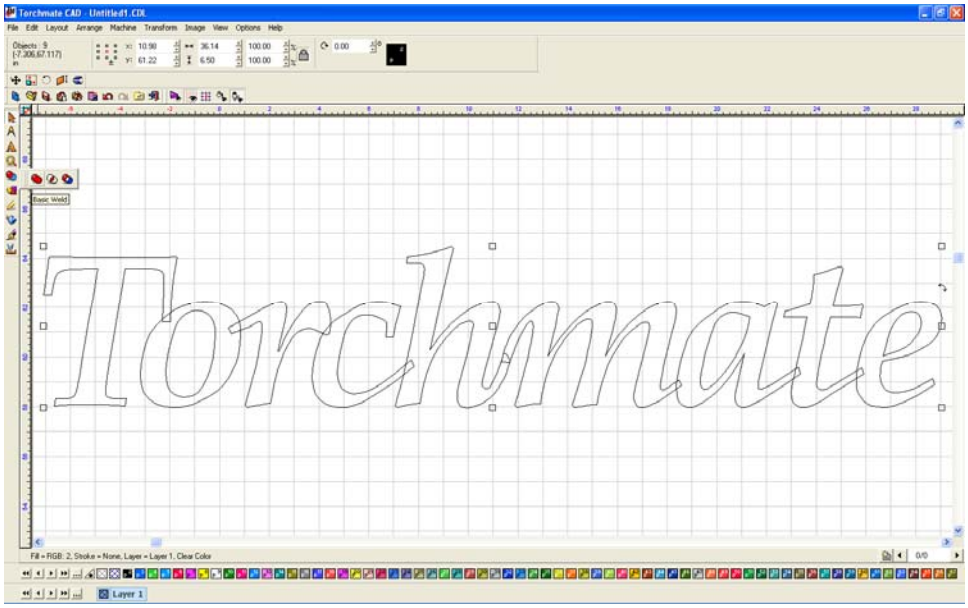
Be sure to overlap all script areas, so that welding can be done later.



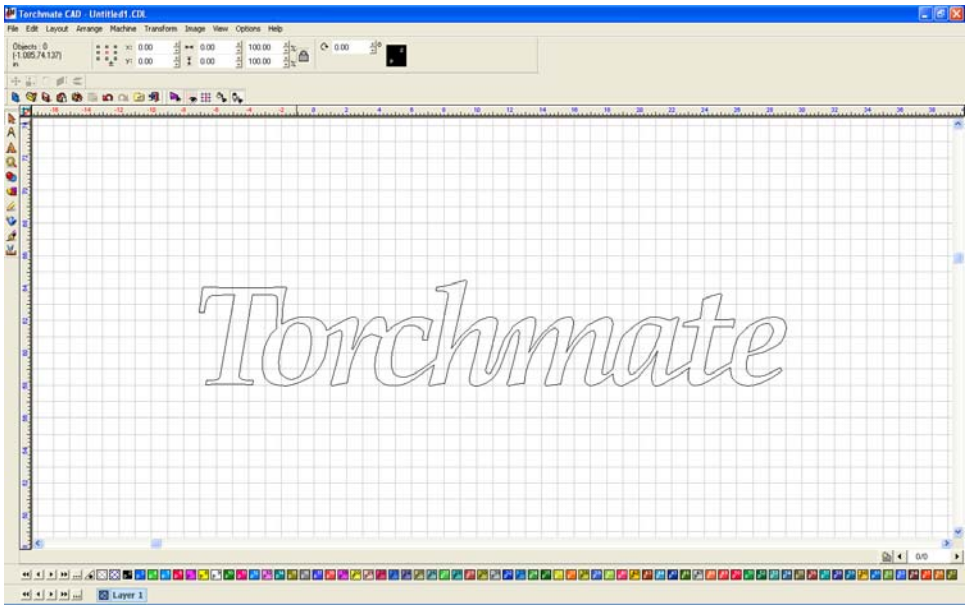
Once all letters overlap the text is ready for welding.



Start by selecting all the letters, and then choose a basic weld.



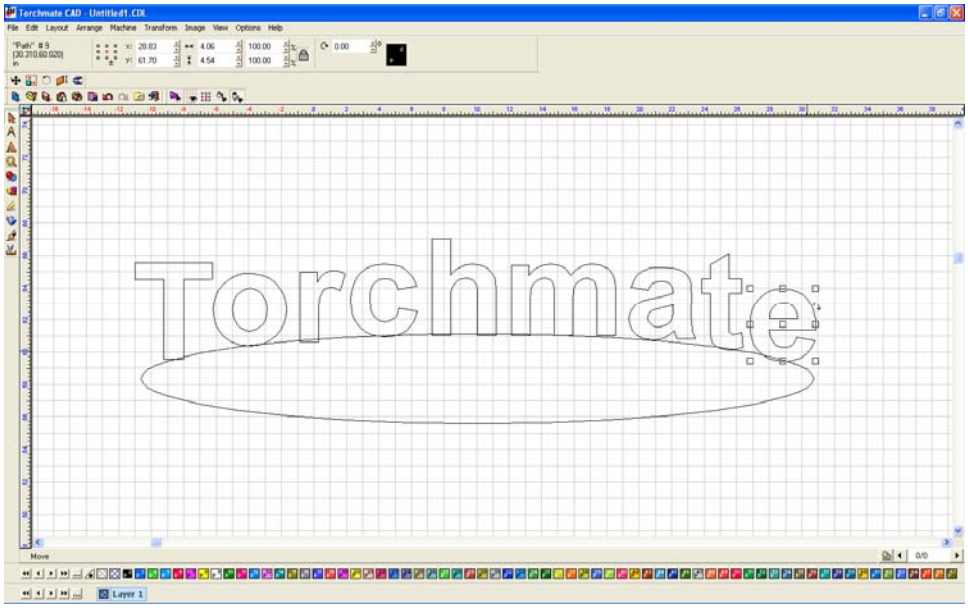
The letters are now all once piece and ready to be cut out. One thing to keep in mind is the thickness of the bridges, if they are too thin when cut out they may bend or break.



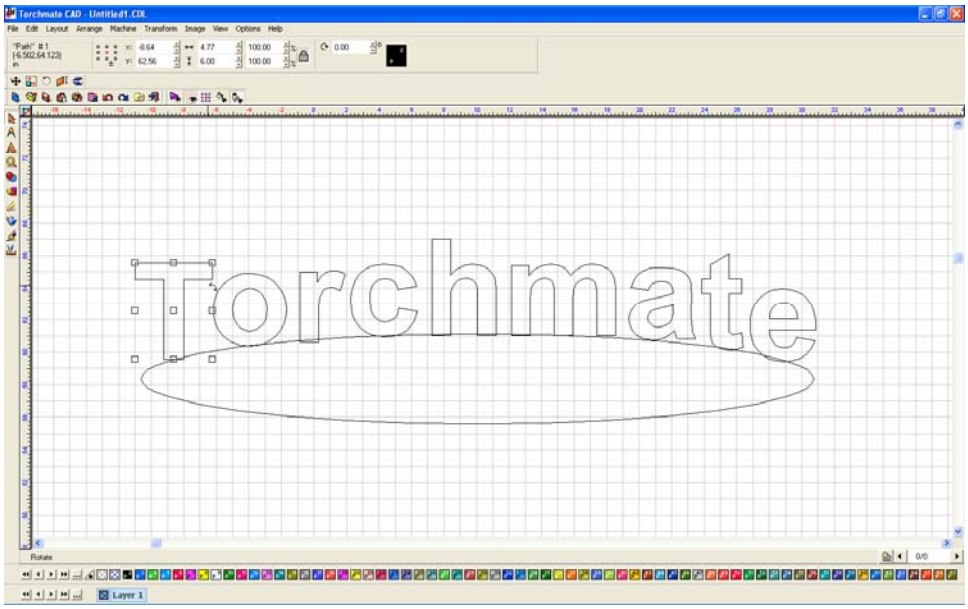
Fitting Text to an Arc:

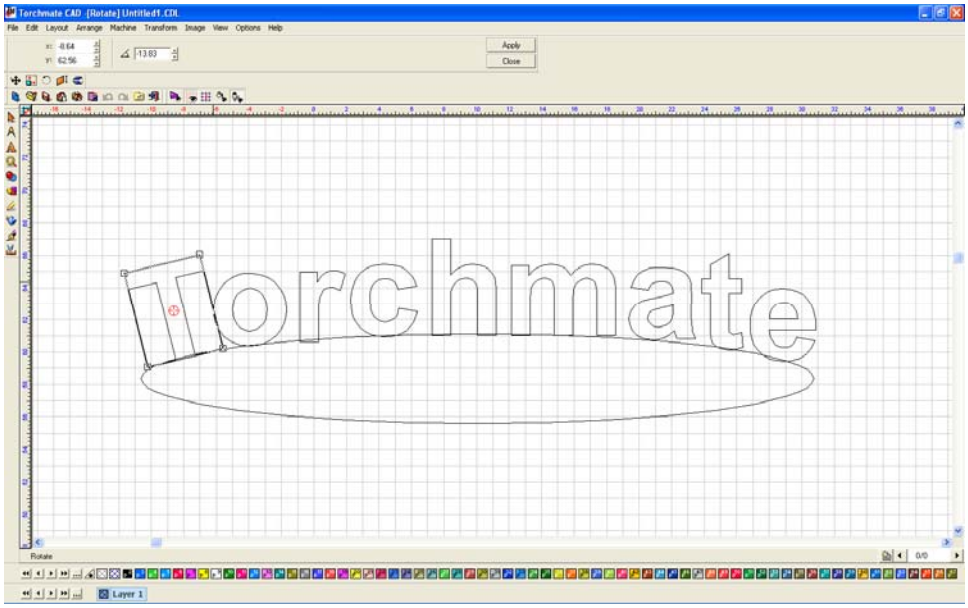
Text can be aligned to an arc to create arches or to follow a specific curve. These features are automatically done with the optional 'Pro Text' module. The instructions below show how to do this function without the additional module.

The first step is to draw a circle or oval that the text needs to be fit around. Once the text is converted to graphics each letter can be moved. Start by positioning the letters around the arc.

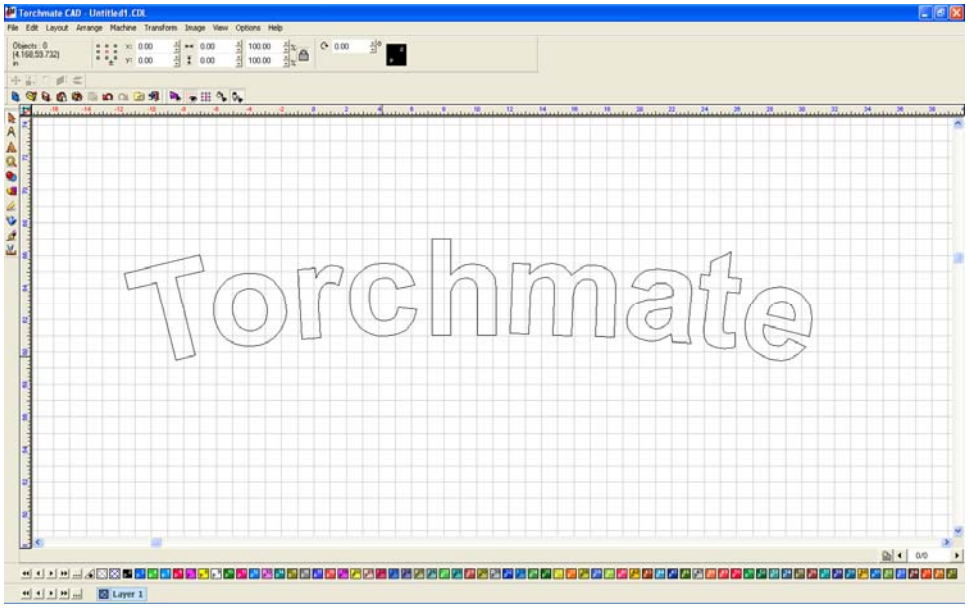


The next step is to rotate each letter. When a letter is selected there are nine boxes around the part as well as a small rotate tab, click on that and as the mouse is moved the letter will rotate.



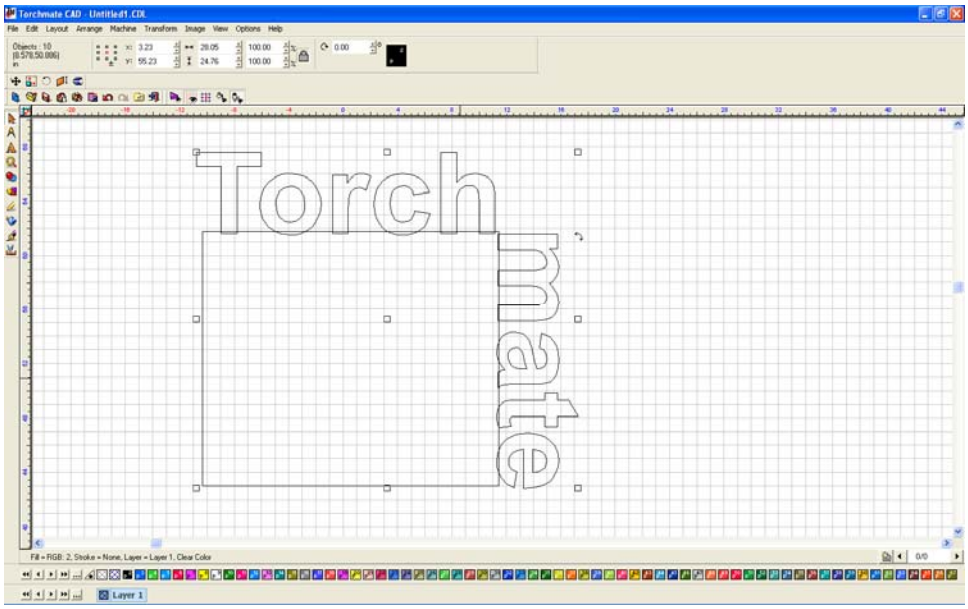


Once all the letters are rotated, delete the arc if it is not needed. This process can be done with shapes as well as arcs, or any shape desired.

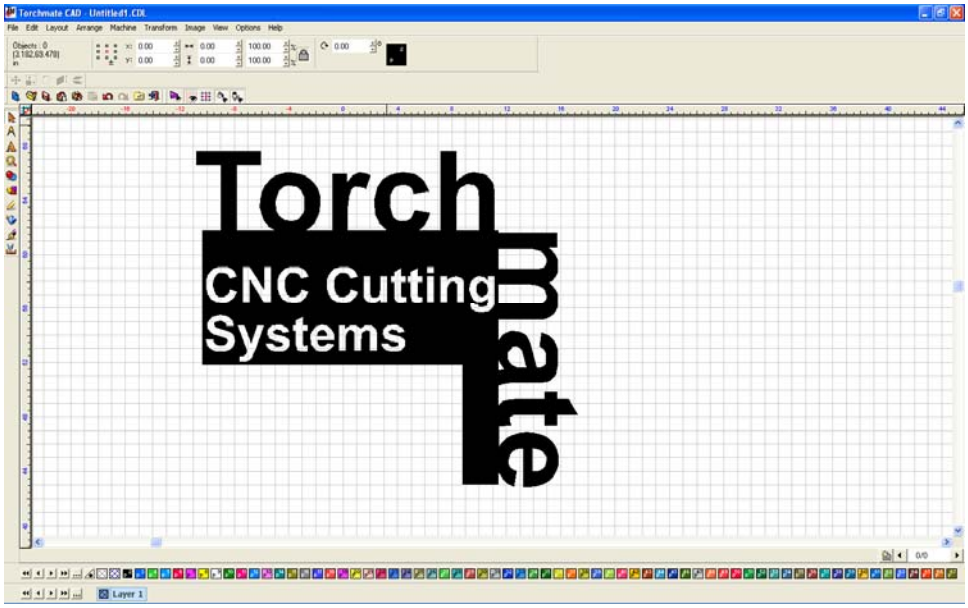


Text Techniques:

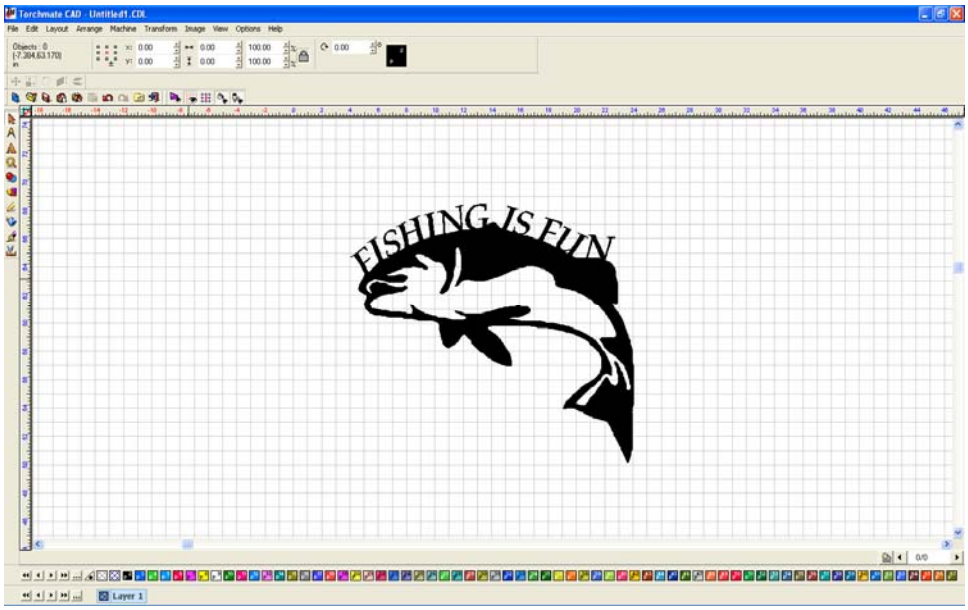
The techniques can be combined to create various effects.

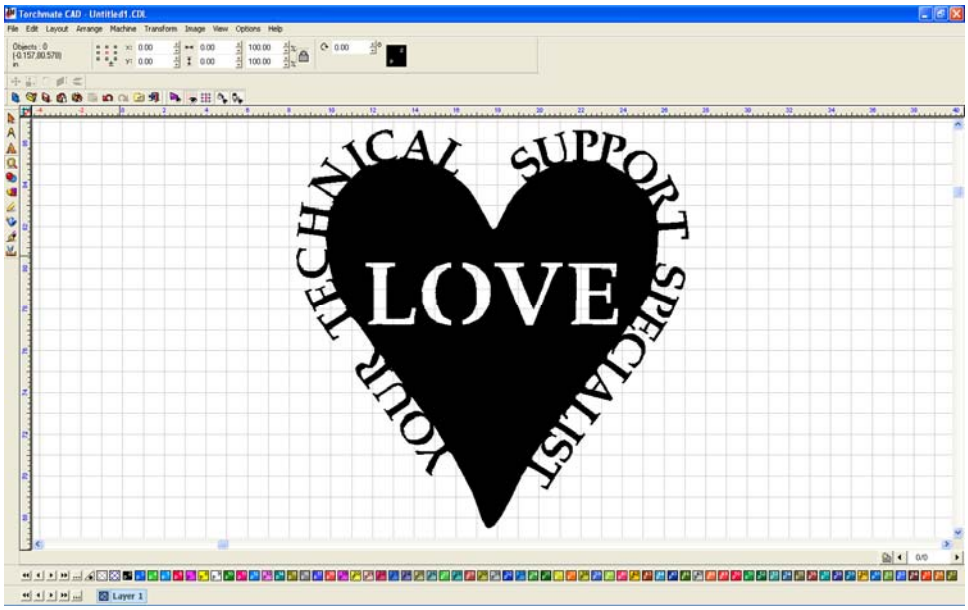


Here is a combination of manipulating text along a path and cutting out text.



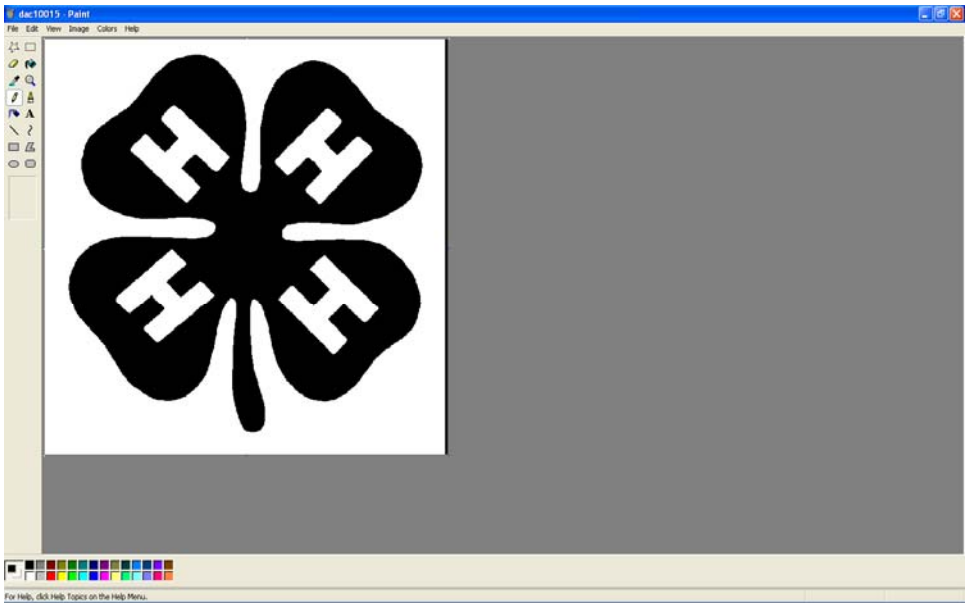
Combining text editing with other skills in Torchmate CAD can allow for creative use of text and plasma cutting.



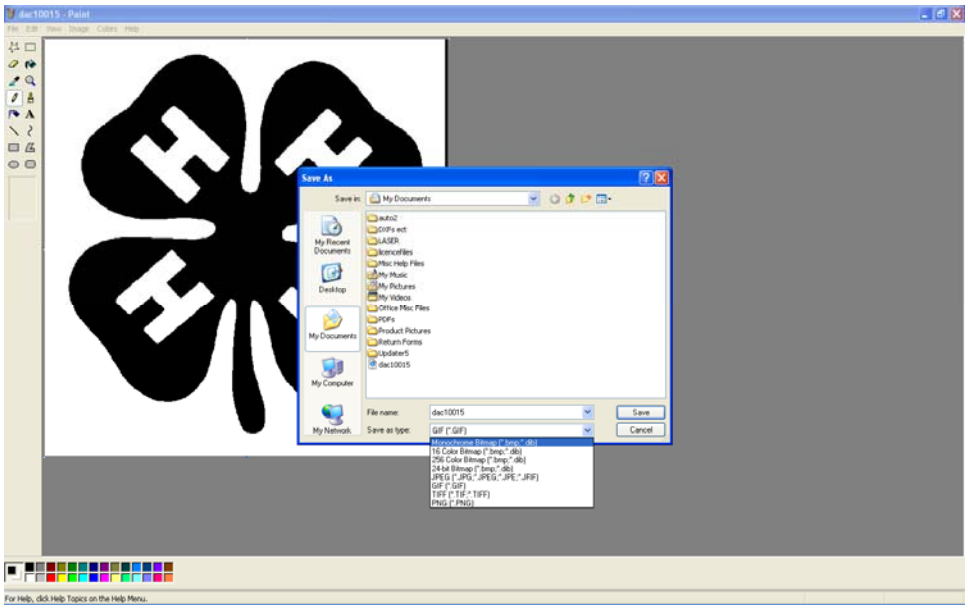


Working with scan tools:

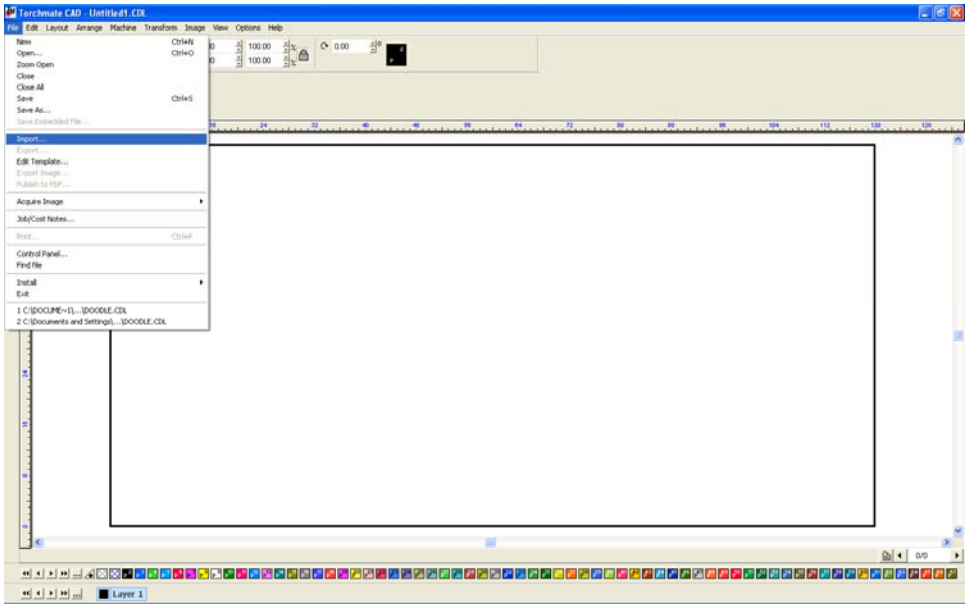
The first step in using a scanned image is to ensure that it is the proper format. To do this open the image in any picture editing program such as Paint.



With the image opened save the file as a monochrome bitmap. This is the file type that Torchmate CAD will recognize for scan tools.

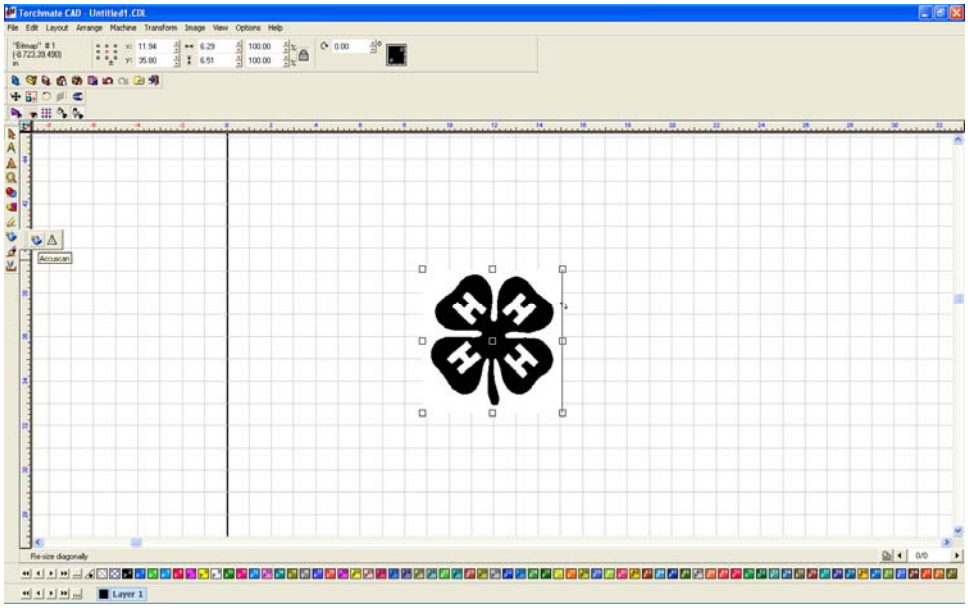


With the file as a monochrome bitmap go to Torchmate CAD, File → Import and select the file.

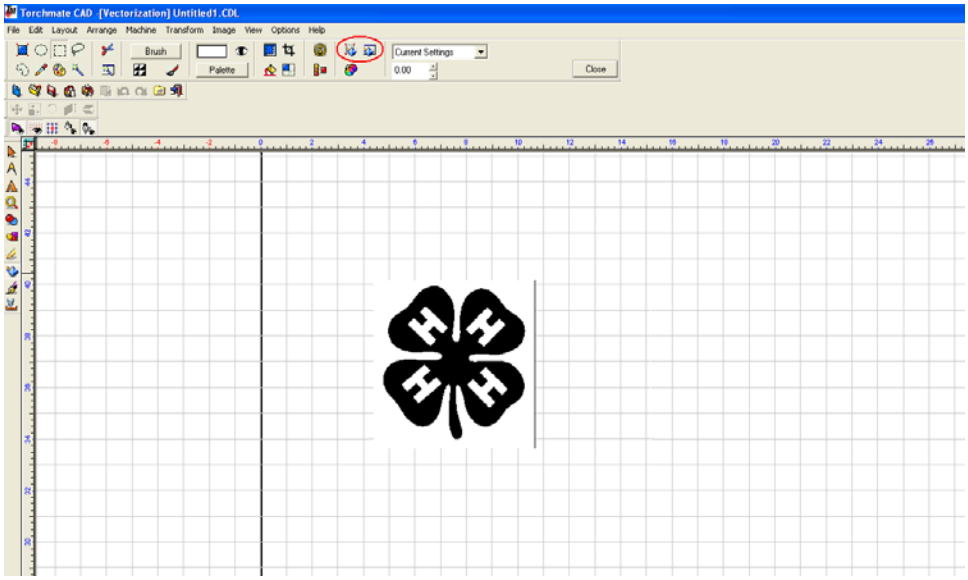


Accuscan:

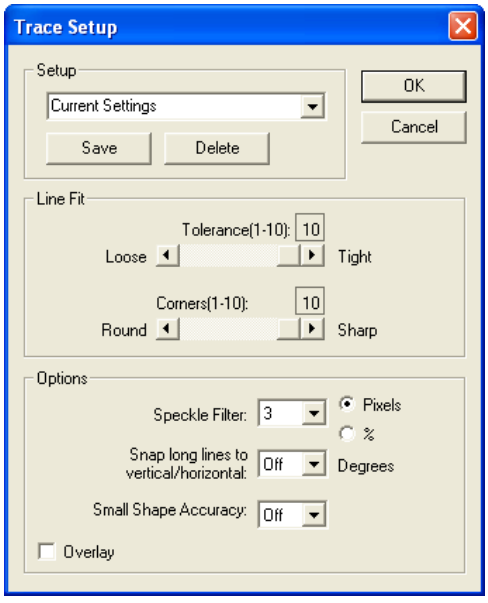
With the image brought in scan tools can now be used. For most applications using "accuscan" will be the best option. What accuscan does is look at an image and it will interpret any black shapes as parts that will be kept as metal and any white parts of the drawing as scrap. Using a "centerline" scan will have the software look the drawing and try to interpret lines, if there was a thin square or box the software may interpret this as a single line not the shape that was wanted.



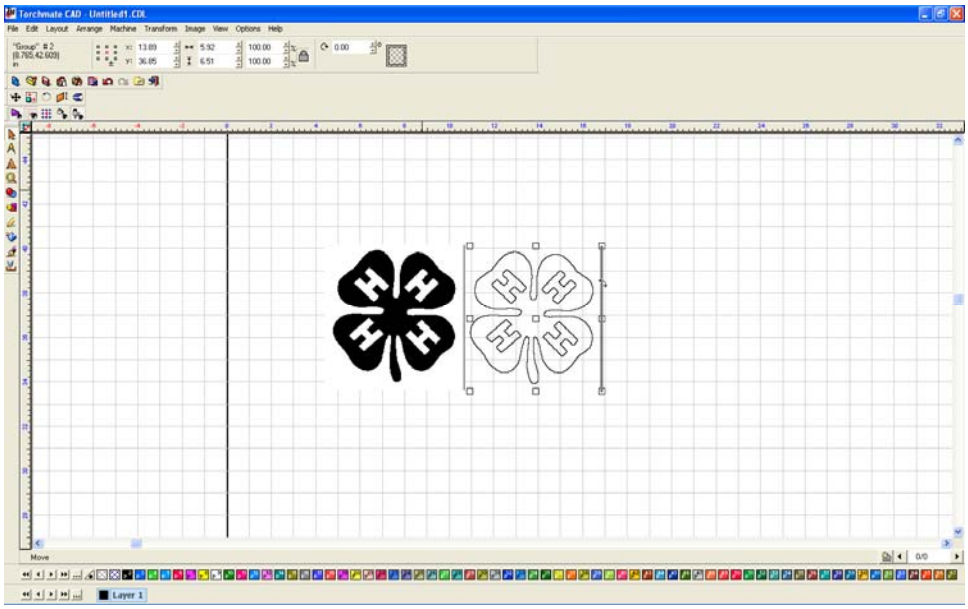
Once accuscan is selected the tool bar on the top will change to the scan tools features. The main features used are two buttons called “trace setup” and “vectorization”. Trace setup specifies various tolerances that the computer takes into account when interpreting the image. Vectorization takes the trace setup numbers and runs a scan on the image creating a line drawing as the computer sees it with the current settings. By varying the settings in the trace setup and running multiple scans each one can be compared side by side to find the best line drawing for a particular image.



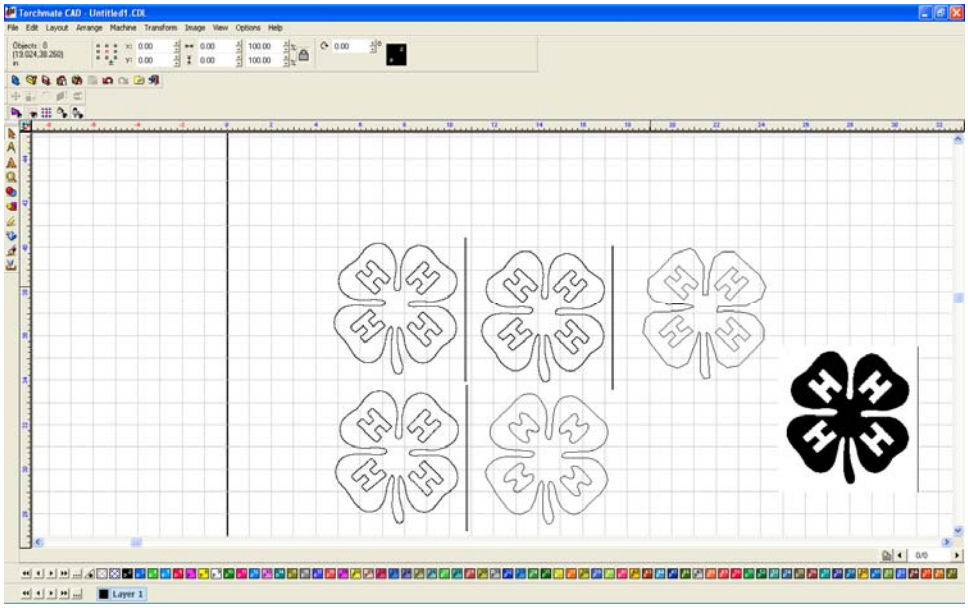
The trace setup for accuscan has a few key settings. The main two are the line fit tolerance and corners. These two features adjust how accurately the computer is trying to follow the image. The additional options effect issues relating to the scanning of an image from a flatbed scanner. If there is significant dust on the image the speckle filter can attempt to clean it up, the snap lines feature adjusts lines if the drawing was crooked in the scanner, and small shape accuracy can compensate for low resolution scans.



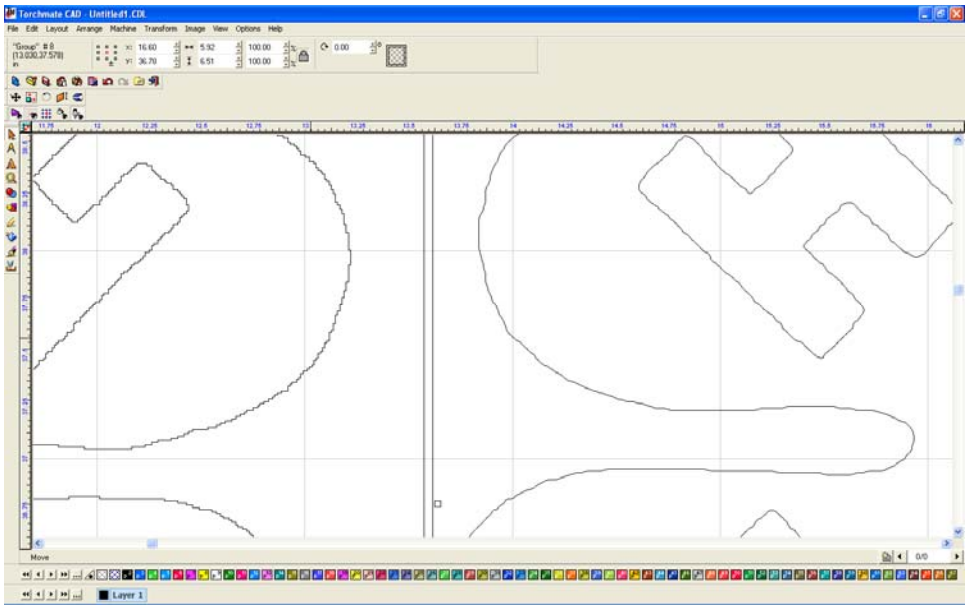
Once the trace setup is done clicking on the vectorization button creates the line trace. There are now two objects on top of one another. Moving one off to the side the two can be compared.



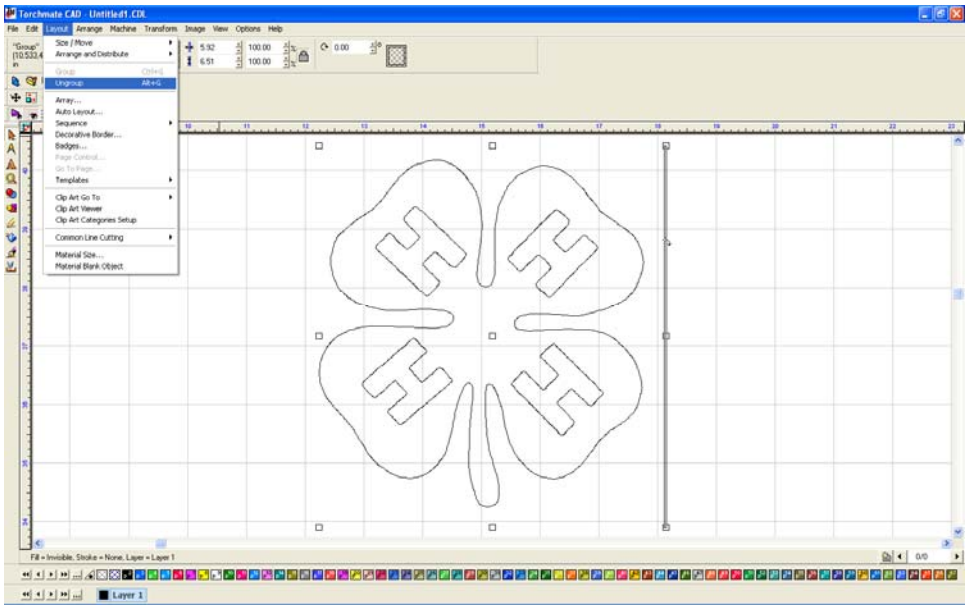
Repeating vectorization multiple times with different trace setups allows the comparison of settings in order to determine the best one for a particular image. Each image brought in may have different trace setups based on the quality of drawing and scan.



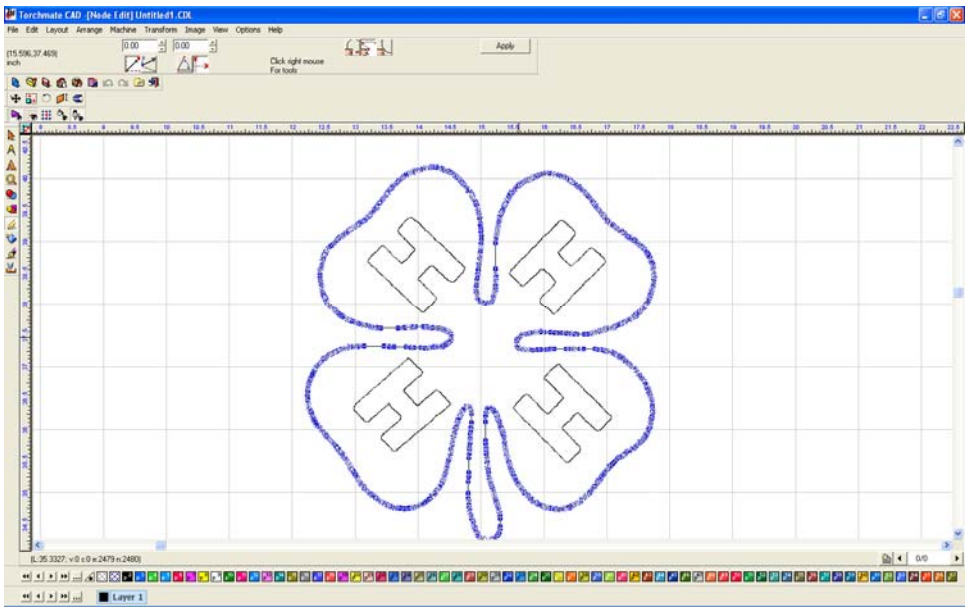
Zooming in reveals small subtleties of the two scans, the left scan is done with 10 and 10 as the line fit tolerances while the right image used 1 and 1.

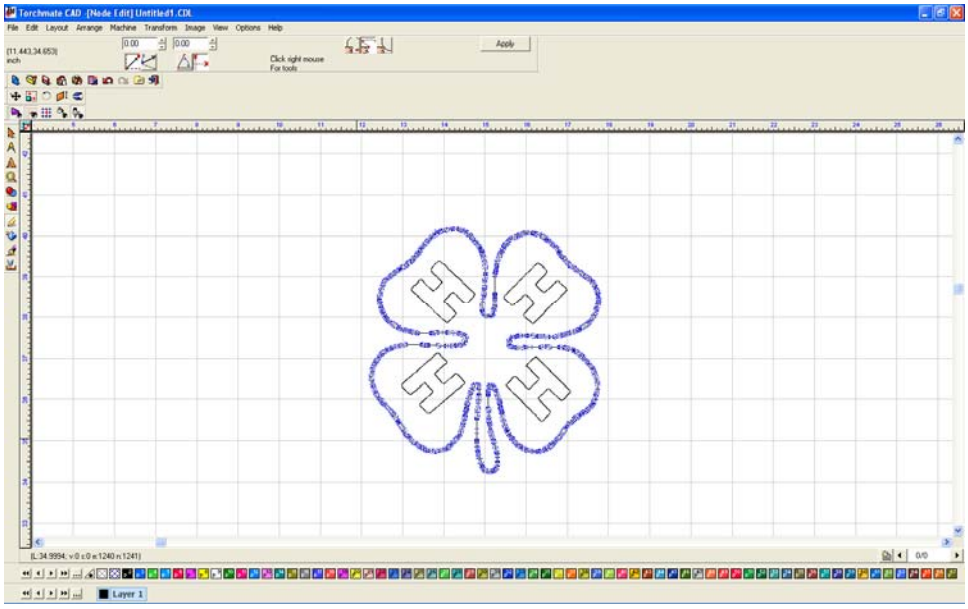


Once the ideal trace has been done the next step is to clean up any necessary defects. The first part of the process is to ungroup the trace so that each piece can be edited individually.

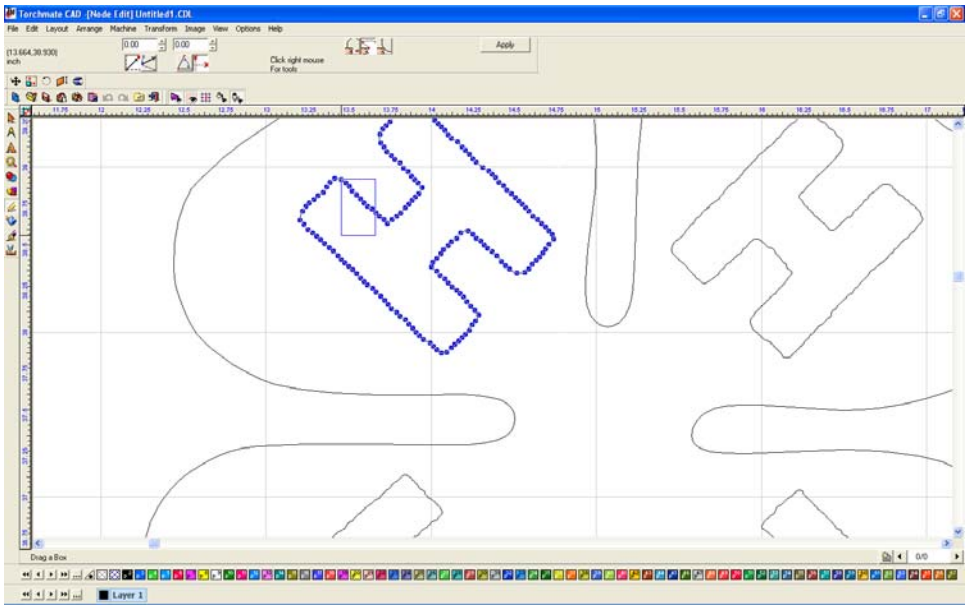


To edit a particular part double click on the line and Torchmate CAD will enter a node editing. Generally when an image is traced there will be an excessive amount of nodes, there a few ways to eliminate them. The most straight forward way is to select the unneeded nodes and press delete, this can work well but if there are hundreds of nodes this can be time consuming. Another method to select nodes to delete is while in node editing select all of the nodes (F3, Ctrl-A, 'Edit' →'Select All', or clicking with the arrow tool and dragging a selection box around all nodes will select all of the nodes). Once all nodes are selected press the 'U' key; what the 'U' does is take all of the nodes selected and select every other one. With every other node selected press delete, this process cleans up a significant amount of nodes in one sweep. Repeating this step a few times can reduce the number of nodes to a manageable amount.

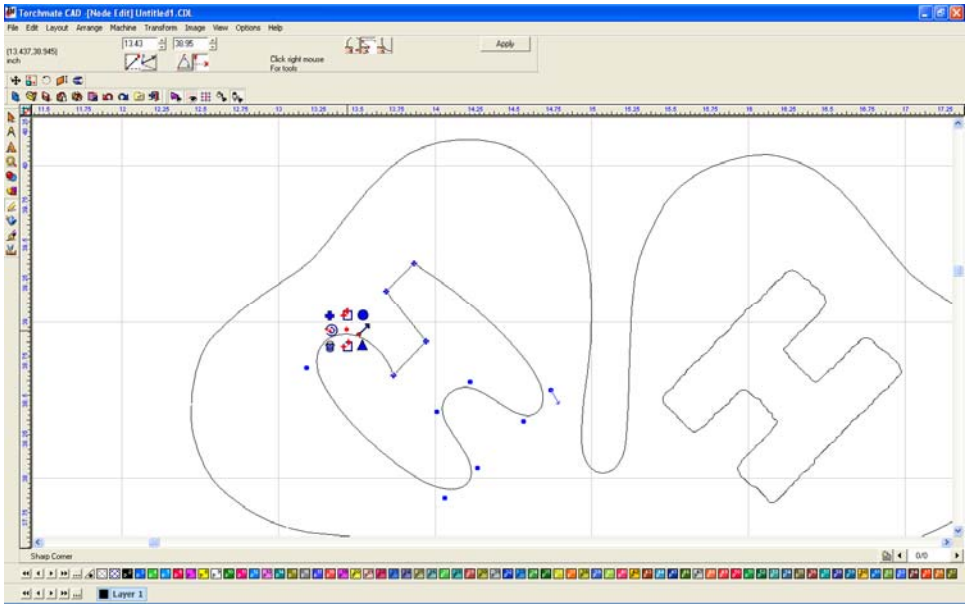




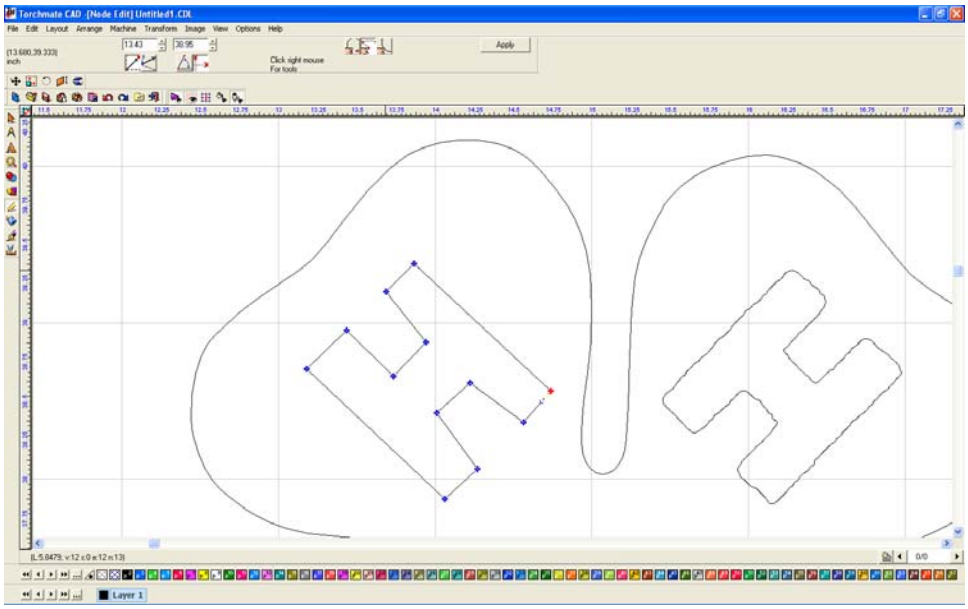
Once the outside is done the letters need attention too. These can be done in the same way selecting nodes and deleting them.



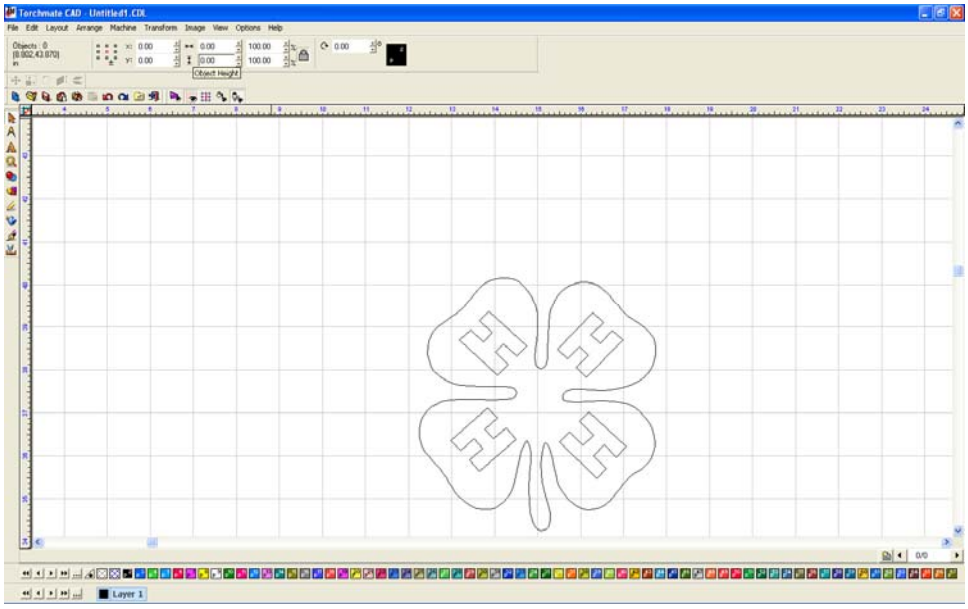
Once the letter is to a point where the only nodes left define the corners the letter may not look correct, almost ballooned sometimes. To correct this issue the nodes need to be changed. There are three types of nodes, a circle node is a rounded node, a cross is a sharp node, and a triangle is a tangent node. With a node selected, right click and move the mouse over the selection wanted and release.



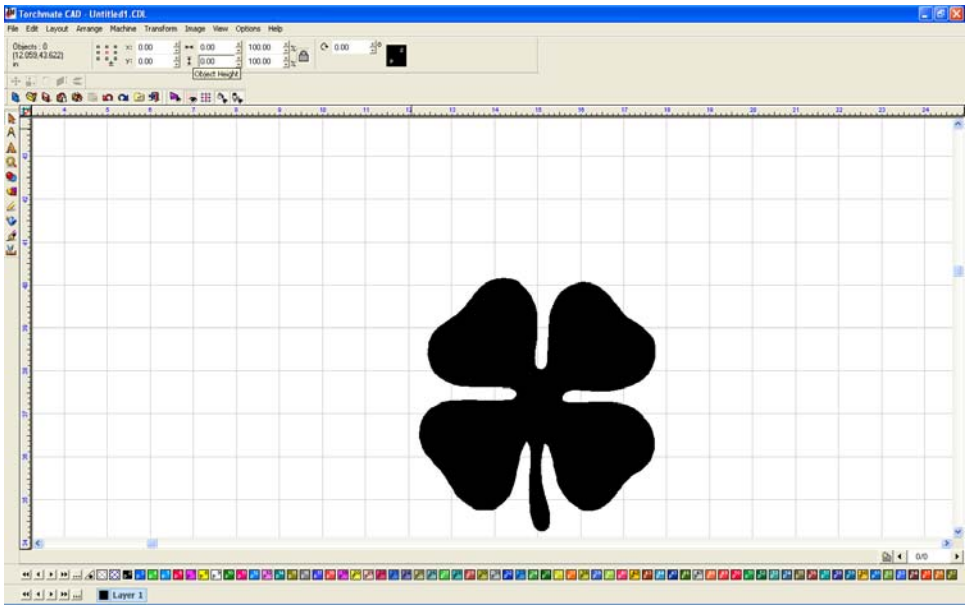
Once the nodes are squared reposition the nodes by selecting one and moving it. Once all the nodes are in the correct position press the apply button.



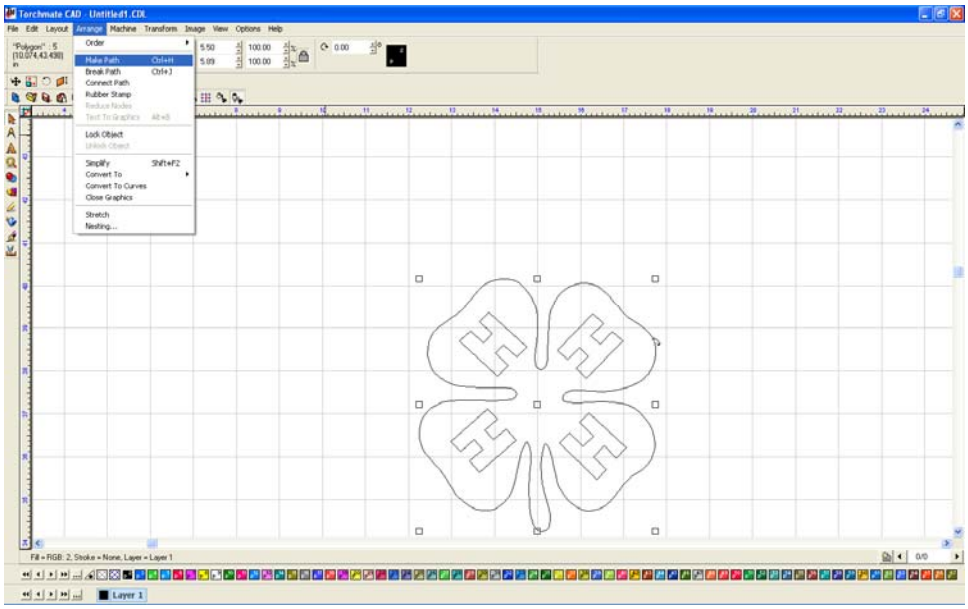
After all node editing is done the drawing is ready to be finalized.



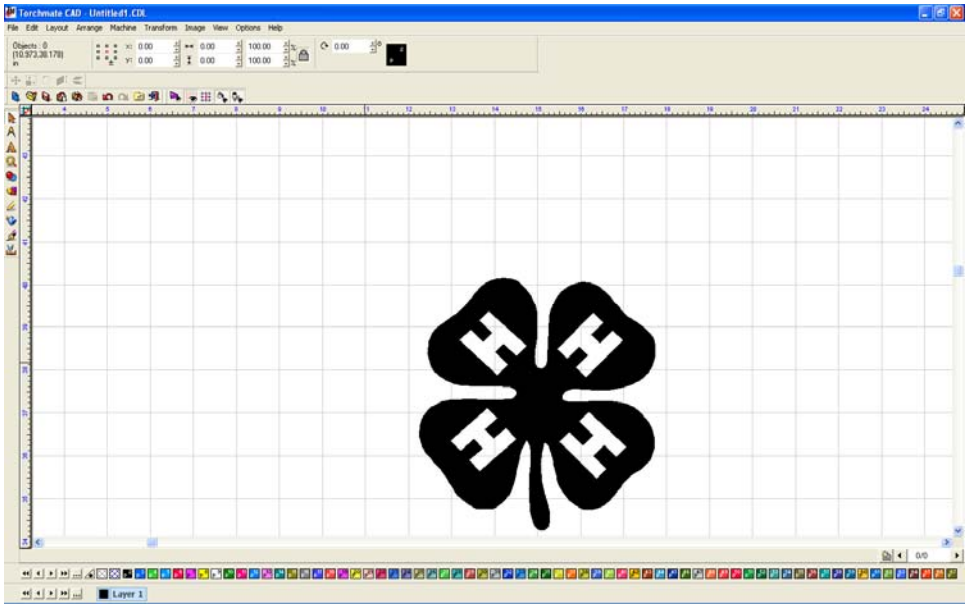
Select ‘show fill’ from the ‘view’ menu, the CAD software is now displaying the object as it sees it. In this case the object is not correct as the letters are not visable.



To remedy the problem select all and go to arrange and make path this tells the computer all that is selected is to be seen as one object.

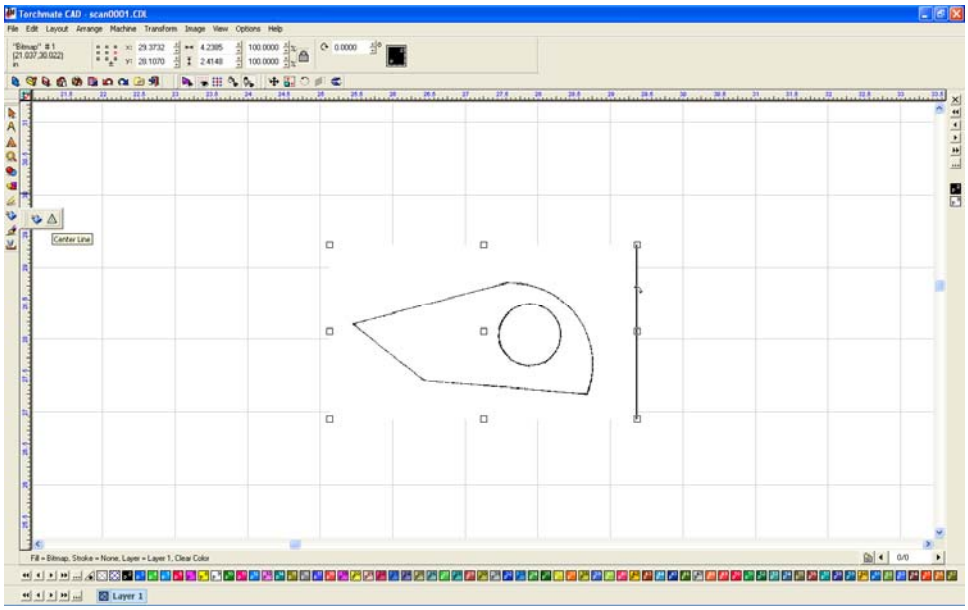


Now with show fill on the object appears as it should and is ready for tool paths or further manipulation.

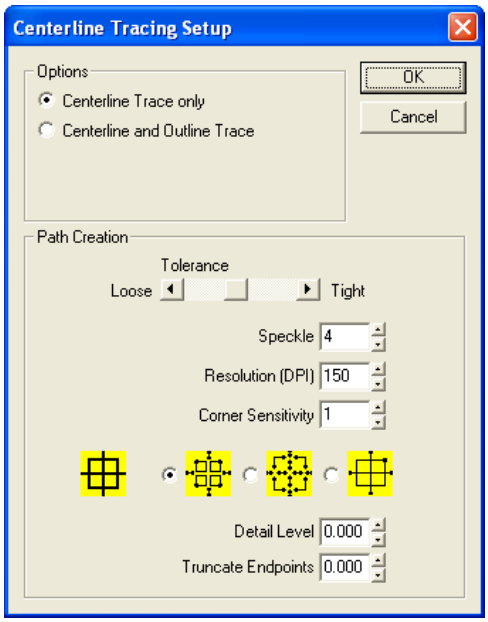


Center Line:

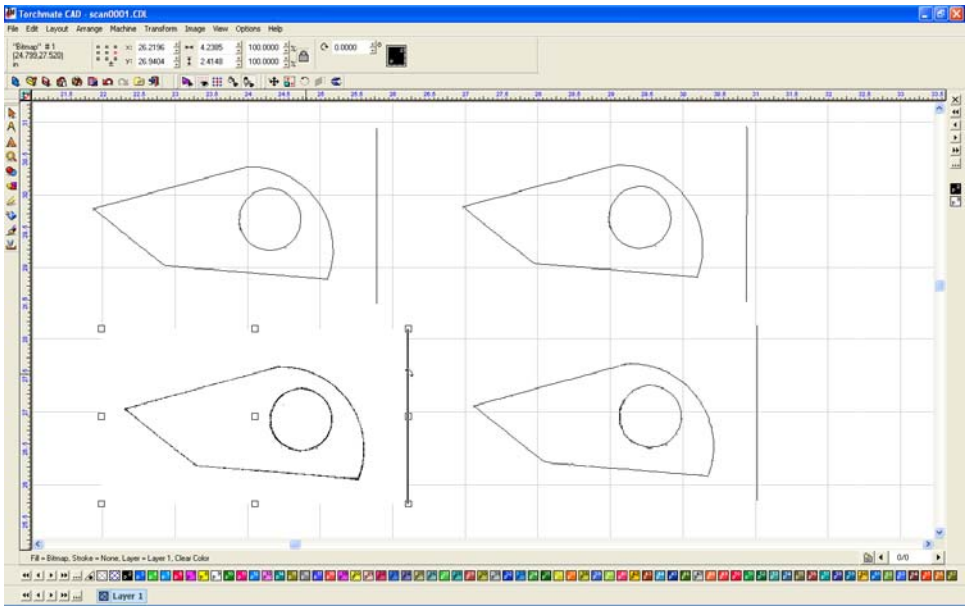
Center Line scanning is used when the scanned image is not shaded in but rather a line drawing.



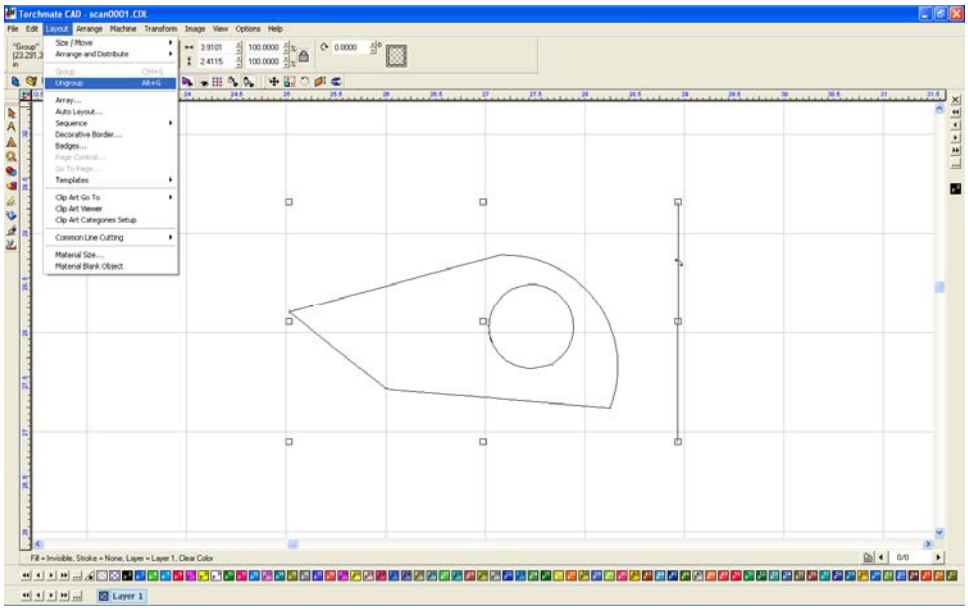
The ‘Tracing Setup’ for center line scanning is similar to accuscan’s. Tolerance will adjust how tightly the image is followed; corner sensitivity will affect how close corners are cut.



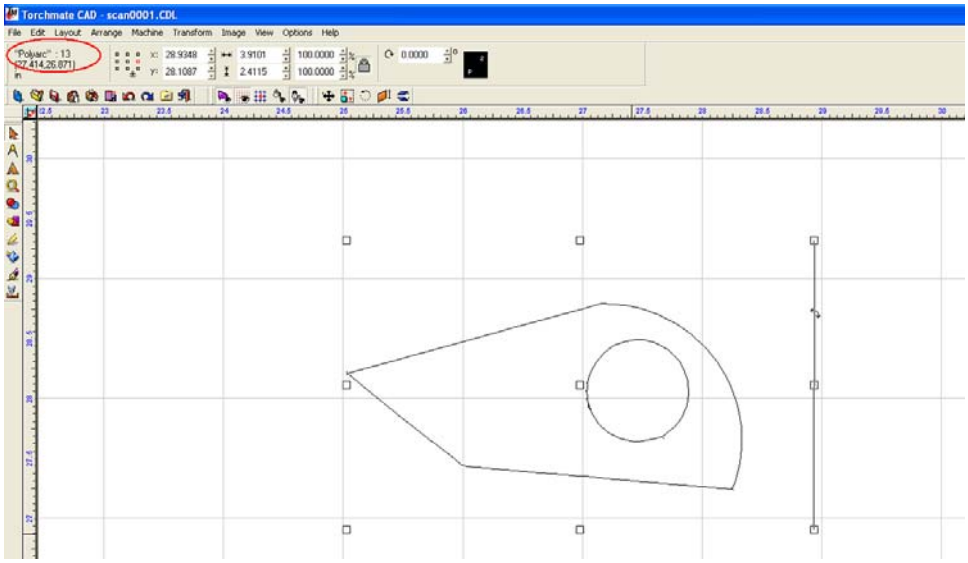
As before multiple scans can be done slightly changing the settings. This method allows for side by side comparison of trace settings to find the ideal one for the particular scan.



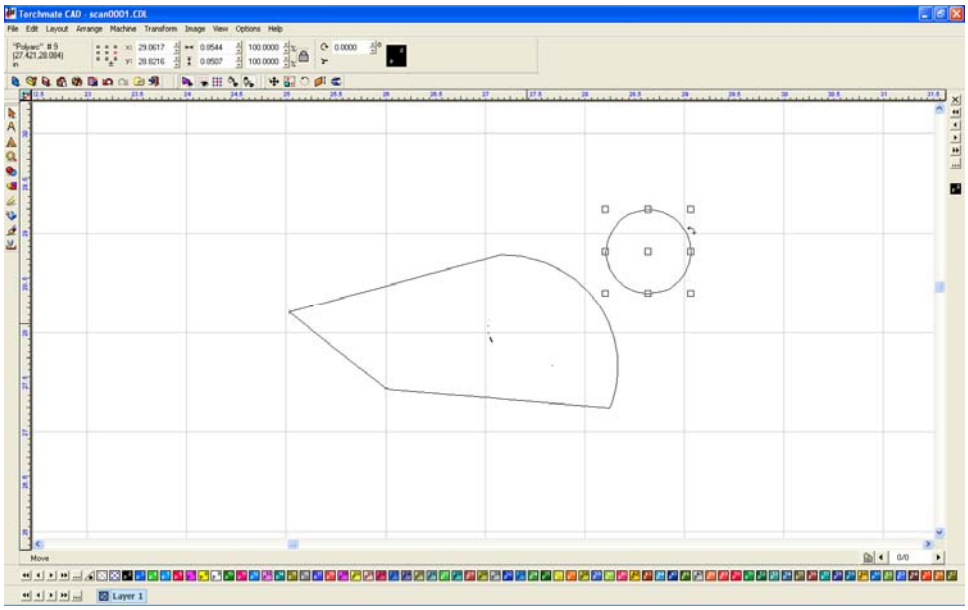
When a trace is done with center line scanning the resulting image will be grouped together. The first step in editing the image is to go to 'Layout' → 'Ungroup'.



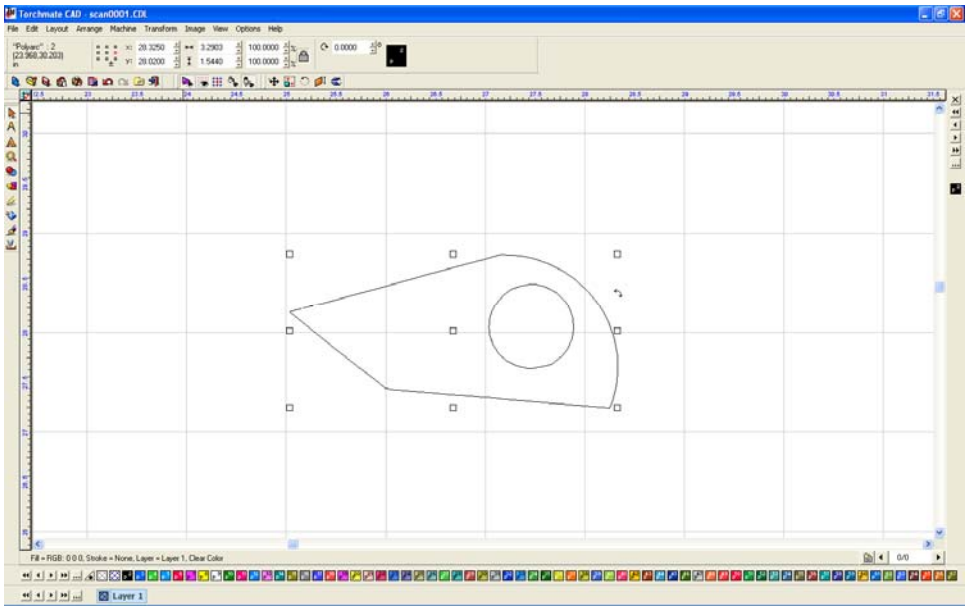
Depending on the quality of the trace the image may be in multiple parts, Torchmate CAD indicates how many objects selected in the upper left corner of the toolbars.



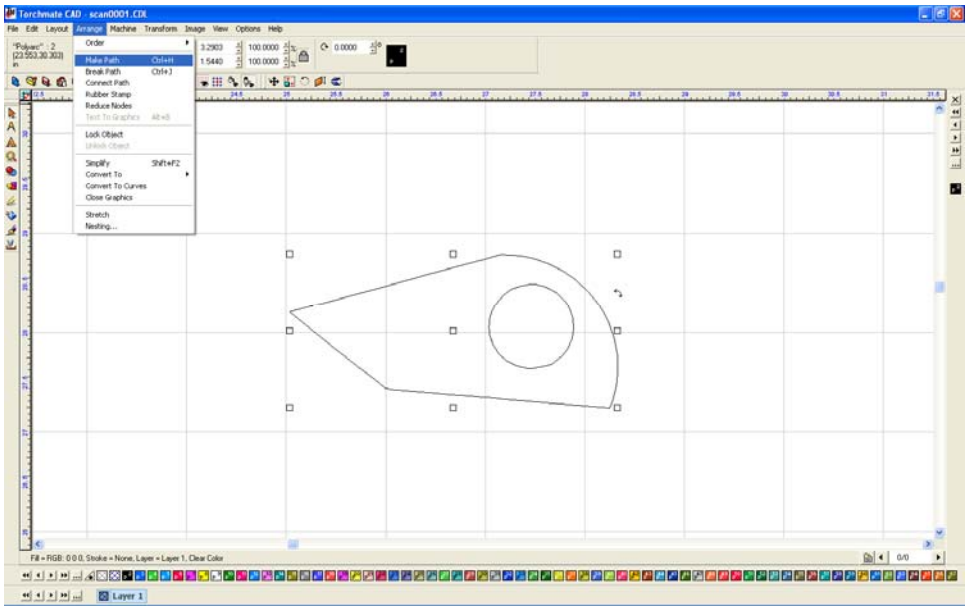
Some of the objects may just be double lines or scribbles, by moving parts away temporarily objects underneath can be seen.



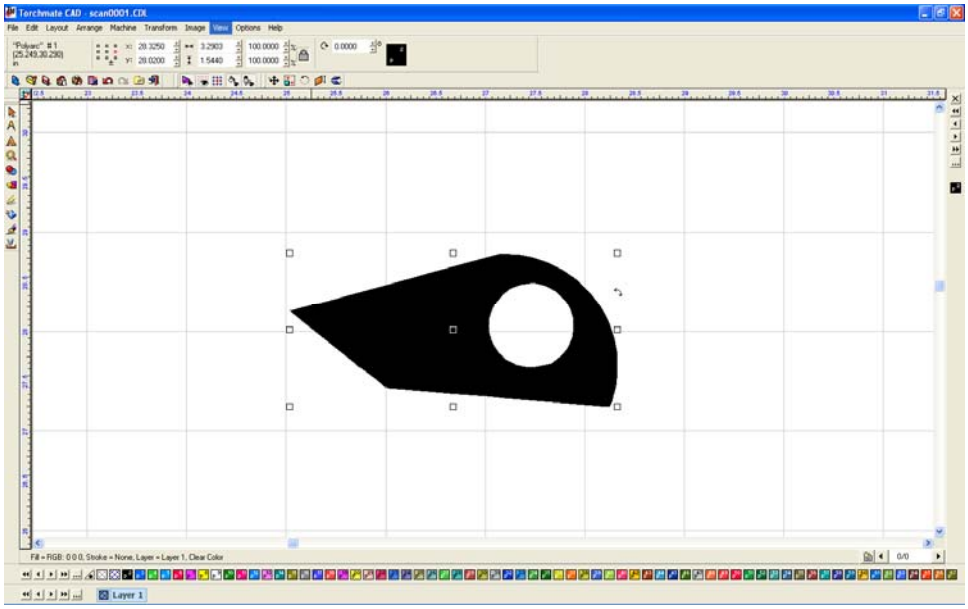
Any fragments or double traces can be deleted until there is just the desired objects left.



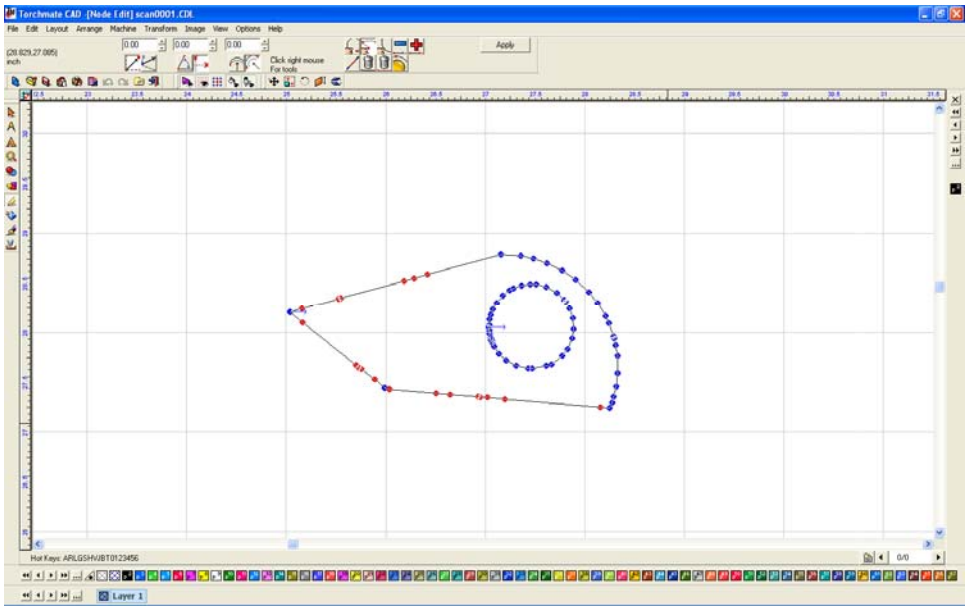
The traced object is now like any other object in Torchmate CAD it can be manipulated, added to, scaled, or tool pathed. To join the two objects as one go to ‘Arrange’ →’Make Path’.



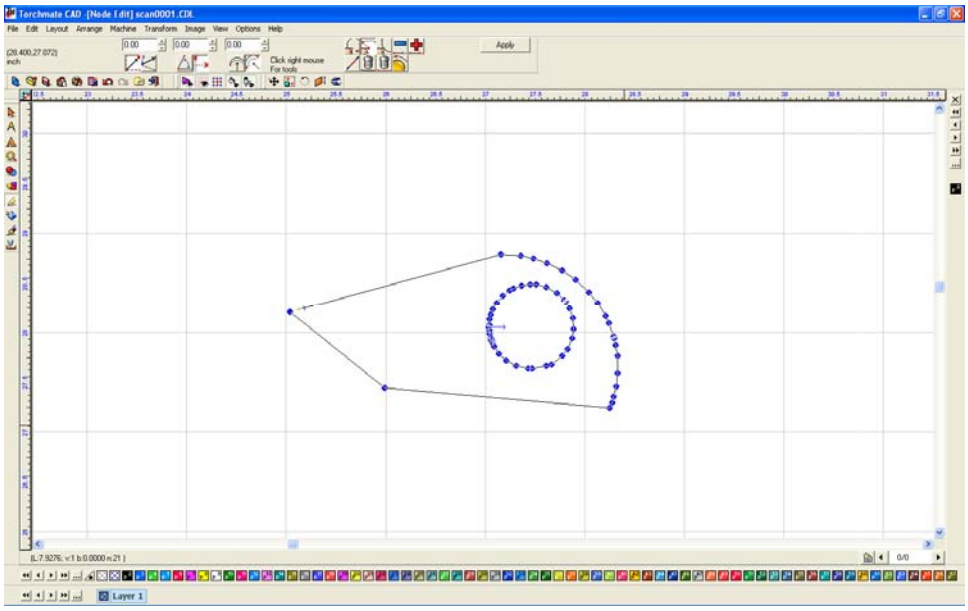
Just like any object ‘Show Fill’ from the ‘View’ menu displays how Torchmate Cad perceives the drawing.



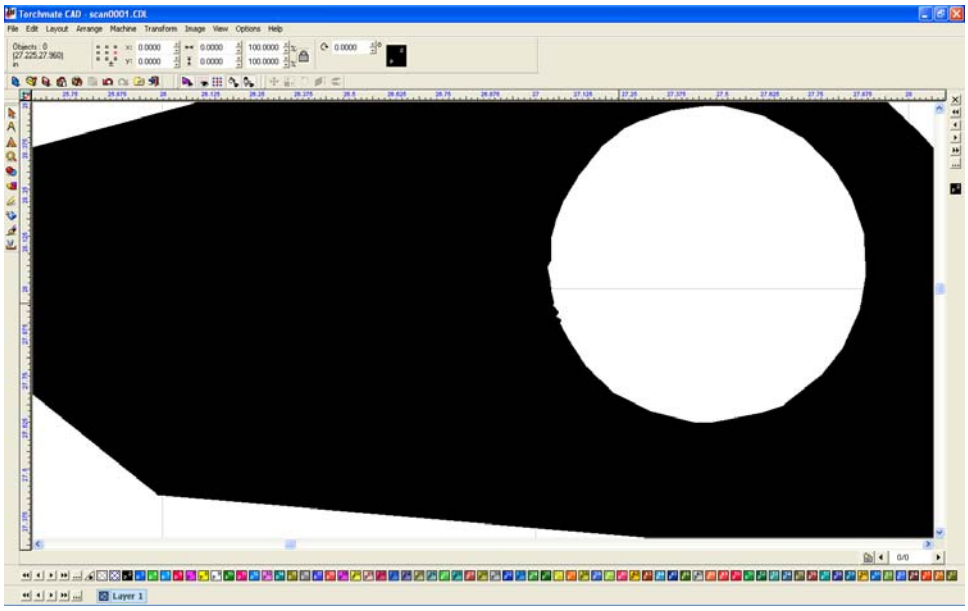
Node editing can be used to clean the object up. In this case selecting nodes that are unnecessary for the overall shape of the object.



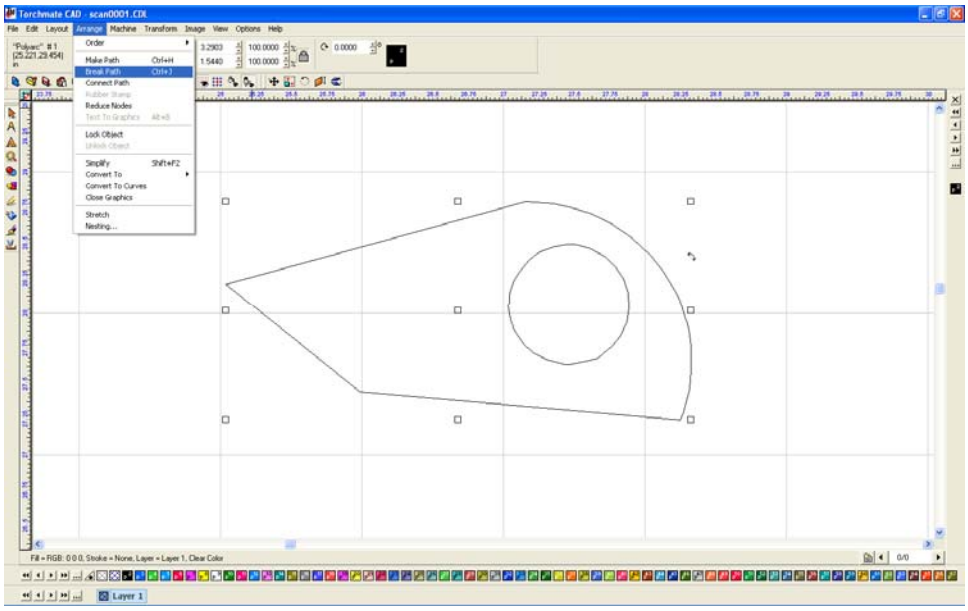
The outside part is now cleaned up, the inside hole can also be cleaned up in node editing.



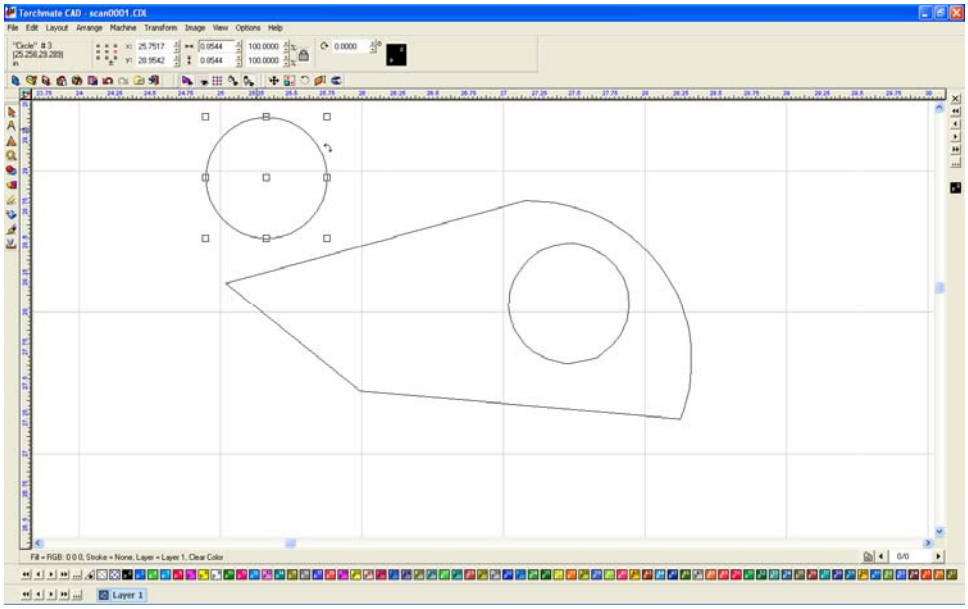
If a hole comes out jagged from the trace node ending can be time consuming. In some cases the easiest option is to redraw the hole with the circle tool.



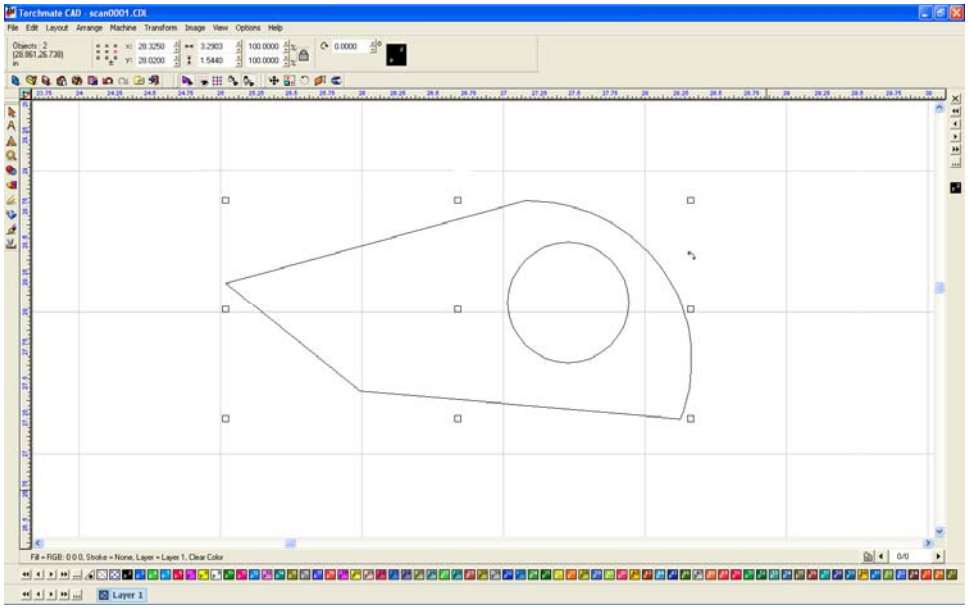
Begin by going to ‘Arrange’ → ‘Break Path’ this separates the hole from the outside part.



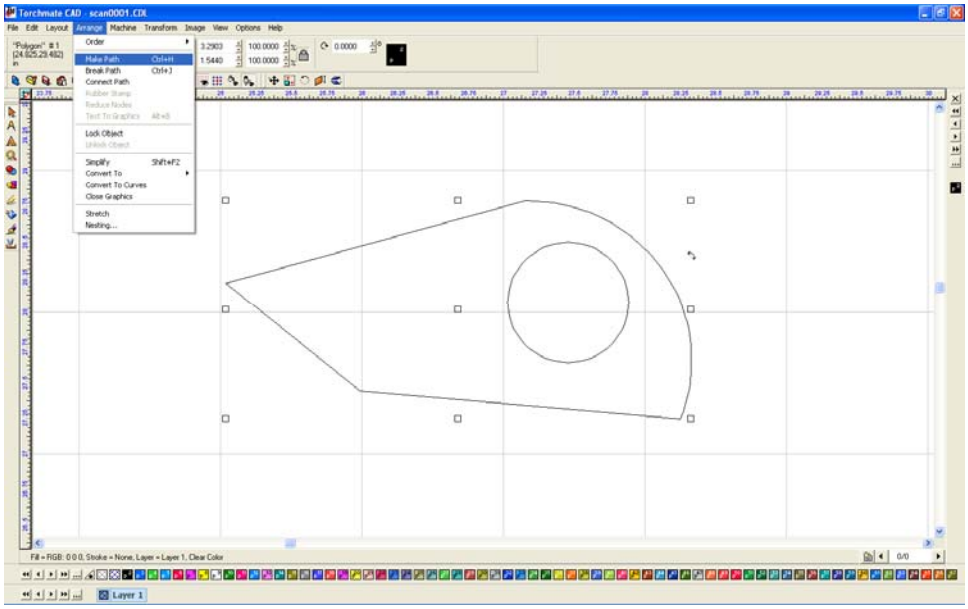
Draw a circle the desired size.



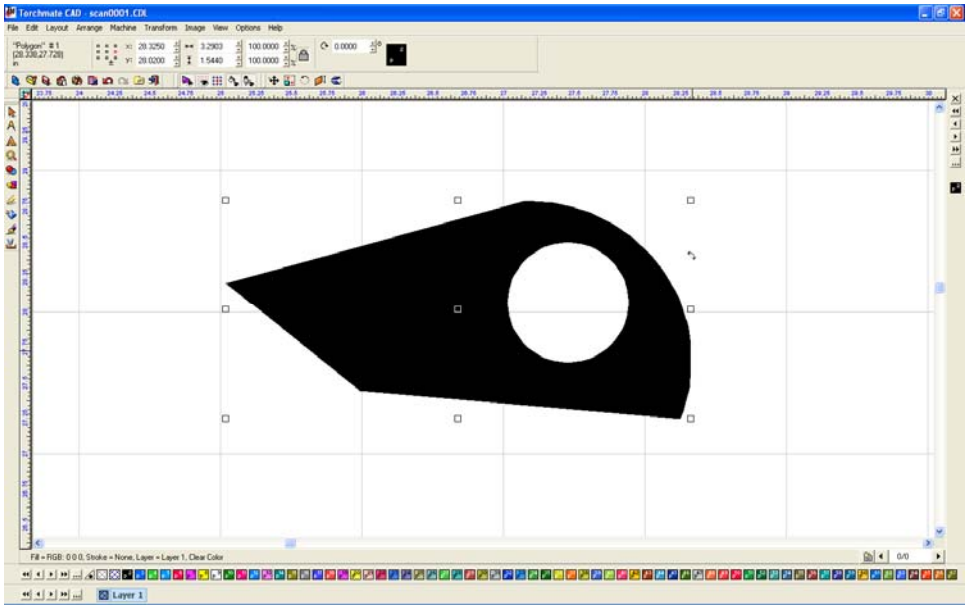
Position the circle where it needs to be and delete the original hole.



Just as before the objects can be made into one part by going to ‘Arrange’ → ‘Make Path’.

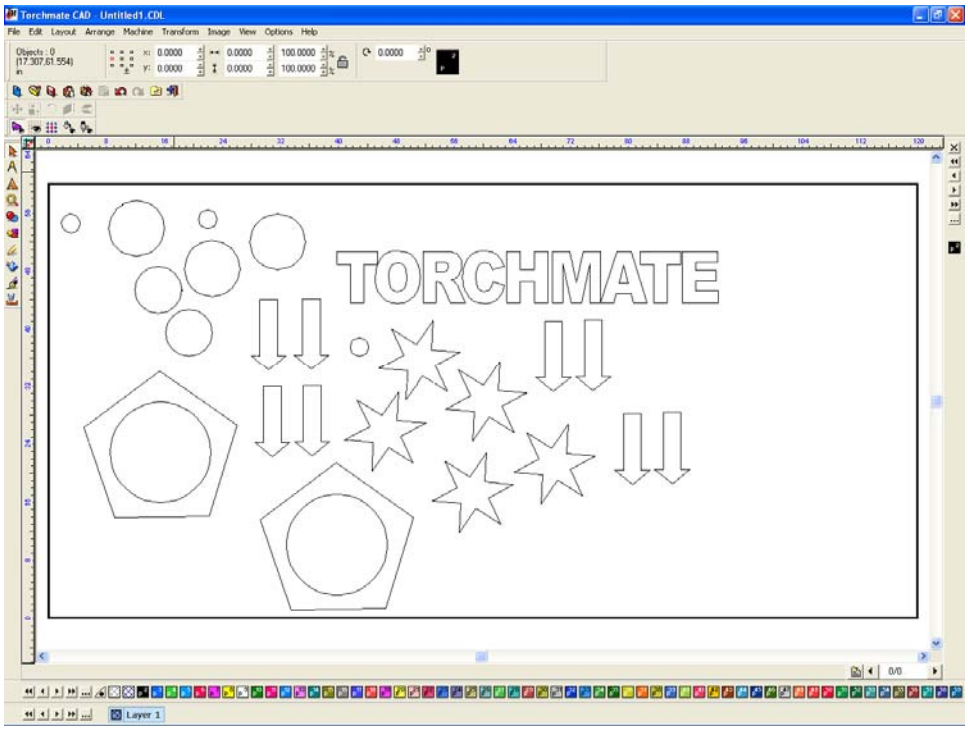


This object can now be duplicated, nested, scaled, or whatever needs to be done before being cut.

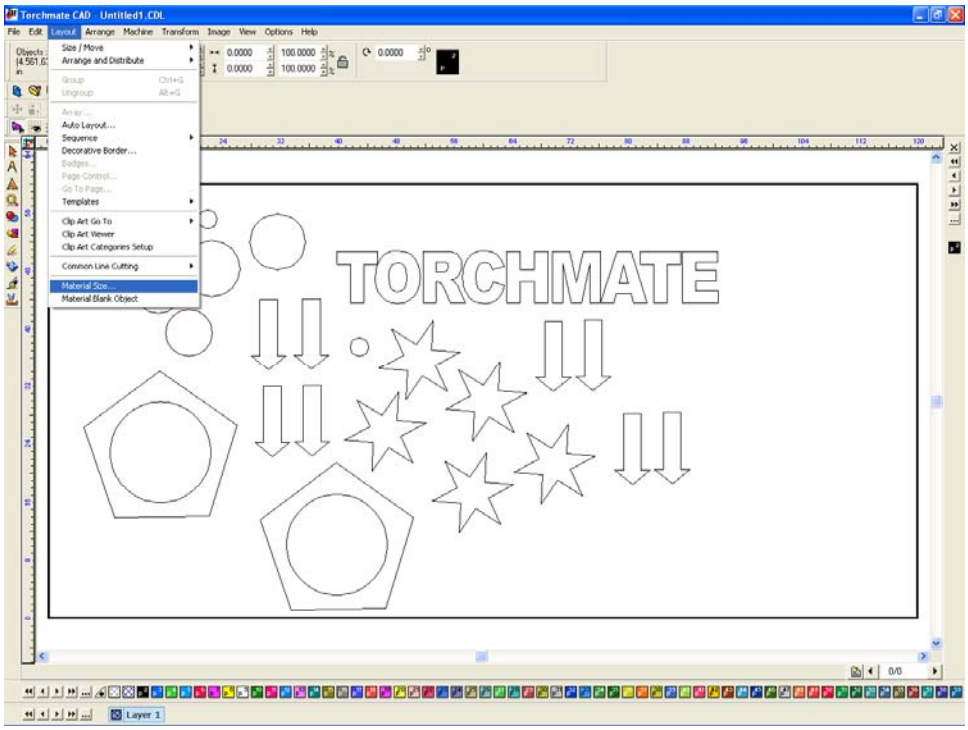


Nesting:

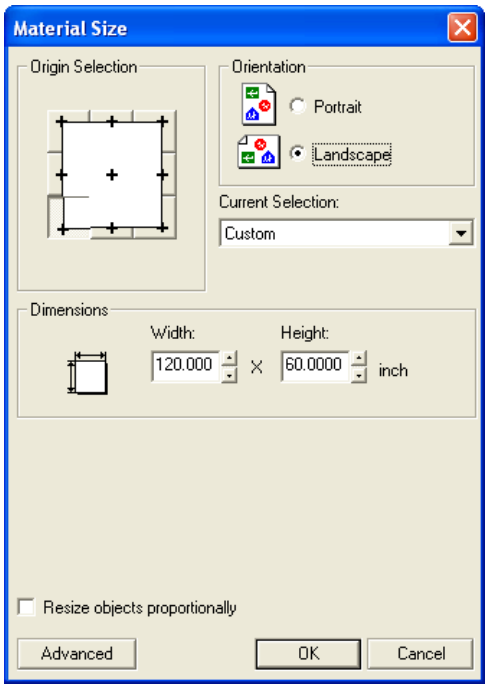
Torchmate CAD allows for multiple parts to automatically be rearranged maximizing the use of material space. This feature is called nesting and can be used on any part drawn in Torchmate CAD.



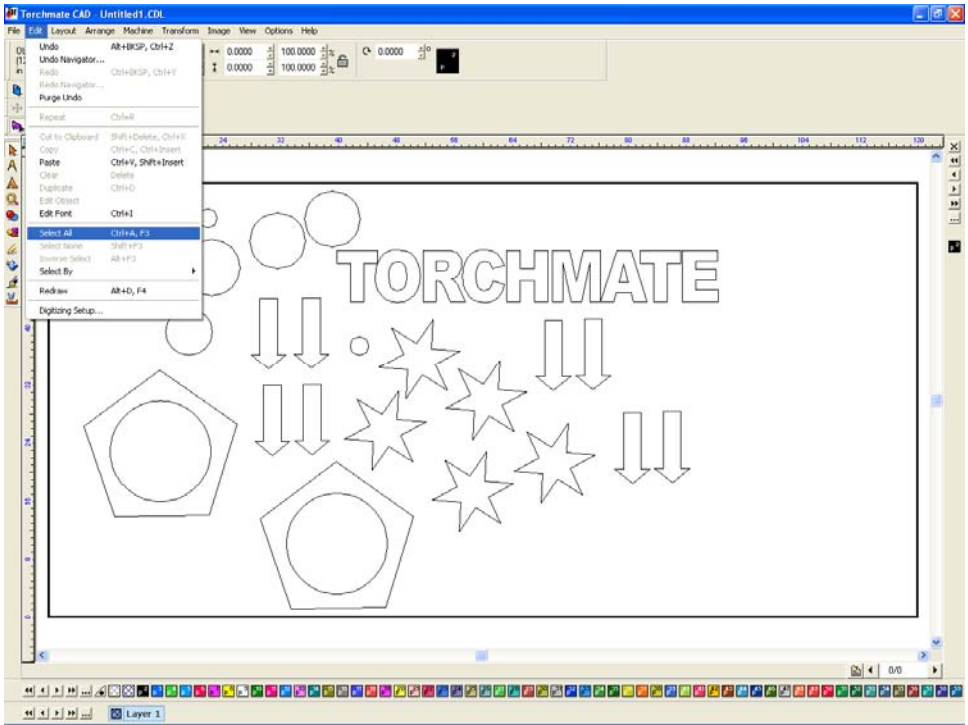
The nesting feature uses a defined material size as its guide to arranging parts.



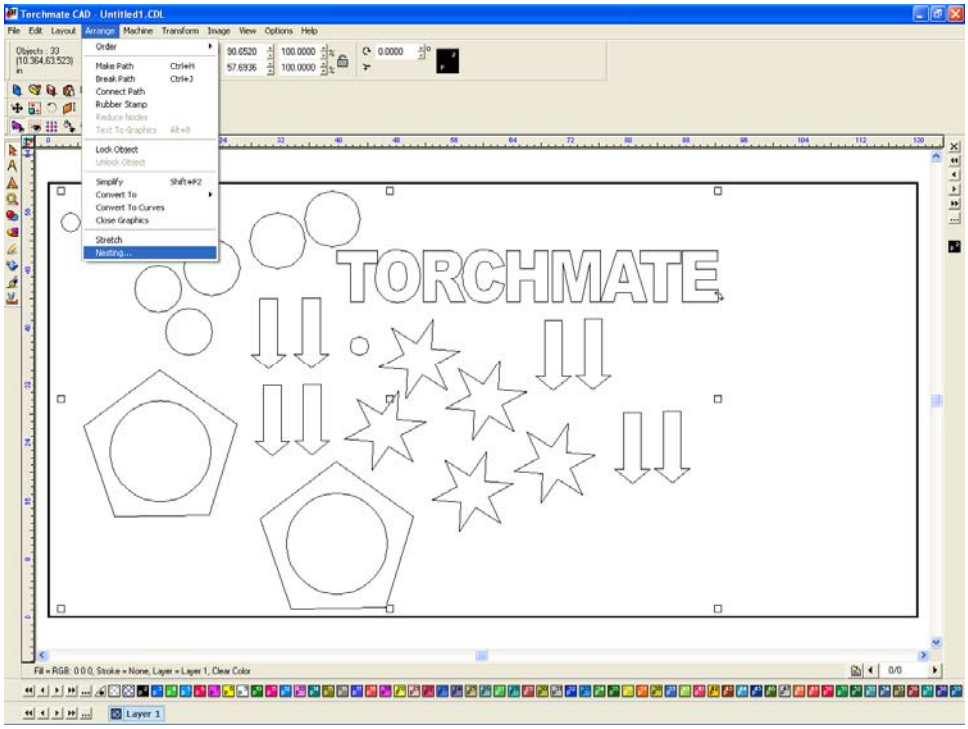
In the ‘Material Size’ window found in the ‘Layout’ menu the size and orientation of the work area can be defined.



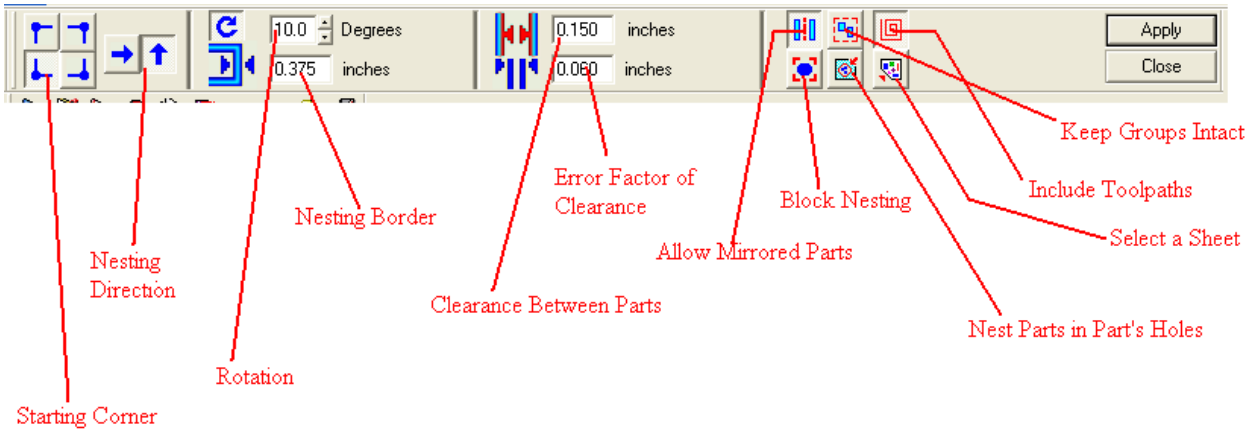
Before nesting can be done all of the objects must be selected, Torchmate CAD will only nest selected objects. Go to ‘Edit’ and select ‘Select All’, CTRL-A and F3 can be used as shortcuts.



To access the nesting options go to ‘Arrange’ and select ‘Nesting’.



Once nesting is selected the top toolbar will change to the nesting options.



Nesting Options:

Starting Corner – Choose a corner that the nesting will originate from.

Nesting Direction – The direction that the nesting will proceed based on the starting corner.

Rotation – Specifies how much each part will be rotated, this is a step rotation so if 60 degrees is specified each part will be turned in 60 degree increments resulting in six possible orientations. The smaller the number is (excluding zero) the more processes the computer will take calculating the nesting.

Nesting Border – This specifies the distance between parts and the edge of the material sheet.

Clearance Between Parts – The minimum distance maintained between parts once arranged.

Error Factor of Clearance – During the nesting process, approximations are made that are based on the magnitude of shapes being nested. Setting a small Error Factor will produce more consistent spacing between shapes but increase the processing time. A larger Error Factor will create less consistent spacing but decrease the processing time of the nesting.

Allow Mirrored Parts – This allows objects to be flipped around to better nest them.

Keep Groups Intact – If this option is off grouped objects will be separated when nested.

Include Toolpaths – Accounts for the diameter of the toolpath in the nesting calculations.

Block Nesting – When selected treats each object as a simple box.

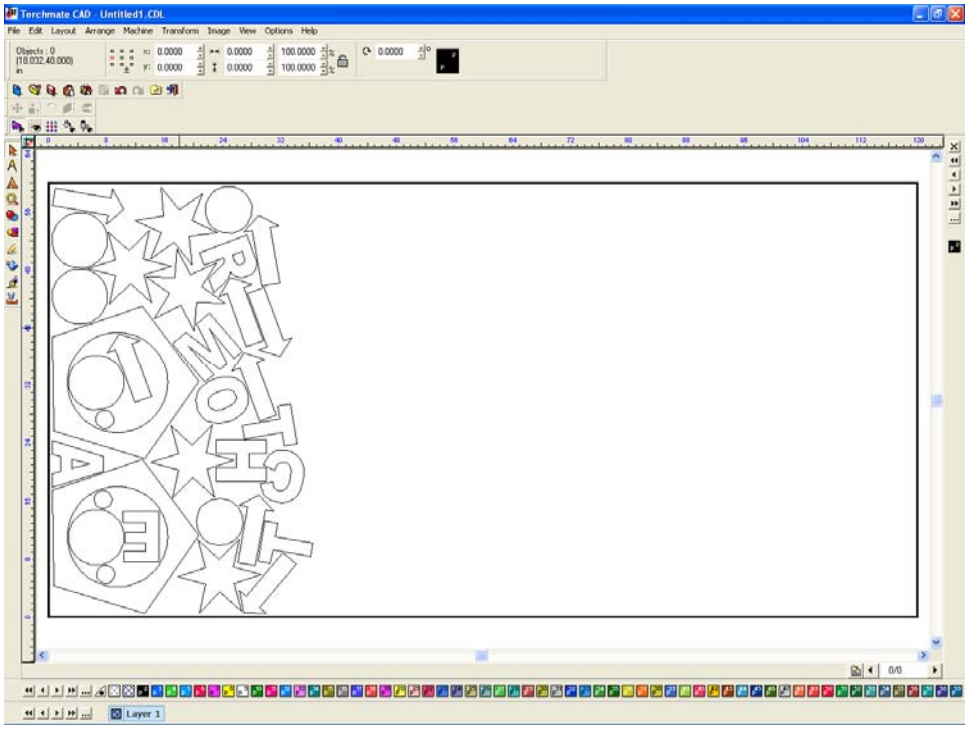
Nest Parts in Part's Holes – This option allows small parts to be moved into larger cutout areas.

Select a Sheet – Allows a specific sheet to be used to put the nesting results on.

Apply – Takes the current settings and begins the nesting process.

Close – Cancels the nesting and returns to the main CAD window, no parts will be nested.

Once apply is selected the nesting process will begin. The process can take several minutes to complete depending on the settings. Nesting can be done as many times as necessary with changes in settings or additional parts added in.



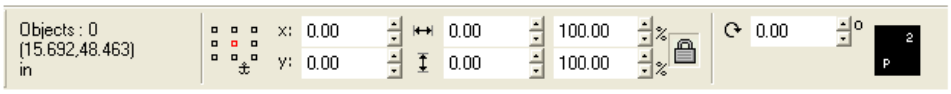
Advanced shape creation:

Torchmate CAD allows for the creation of unique and creative parts. Certain parts require going beyond the basic skills and using all or most of Torchmate CAD's features.

The 'Shape Tools' allow for simple shapes to be made, the simple shapes can be manipulated in order to create more advanced parts.

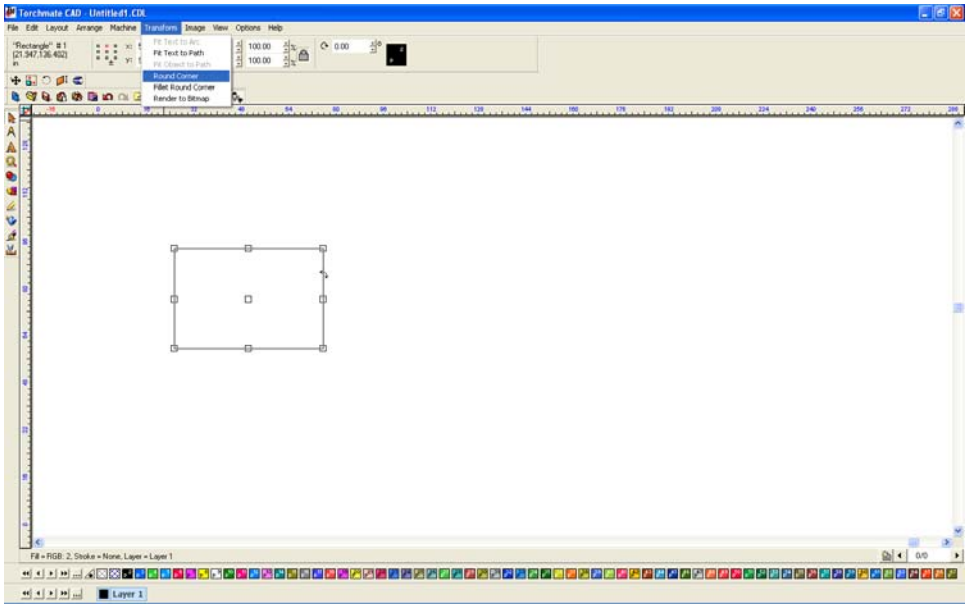
Any shape created with the shape tools will have two different tool bars along the top of the CAD window that can be brought up. The standard shape tool bar is present for any shape or imported drawing when first selected. This tool bar allows the position, scale, and rotation to be changed. The shape specific tool bar is accessed by double clicking on a selected shape. The shape tool bar is only accessible for shapes created from the 'Shape Tools' menu, double clicking for all other shapes will enter the node editing mode.

Standard Shape Tool Bar

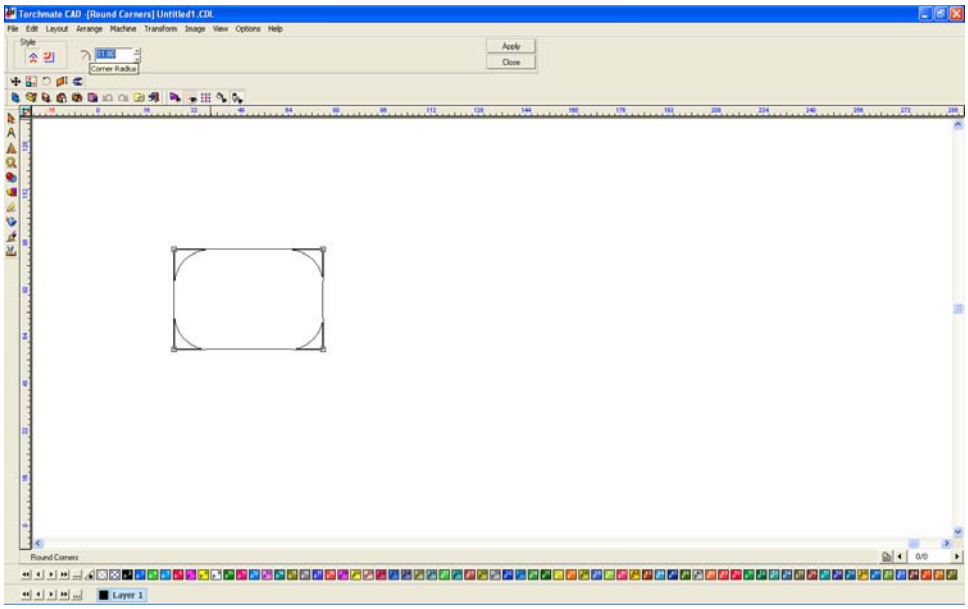


Shape Specific Tool Bar

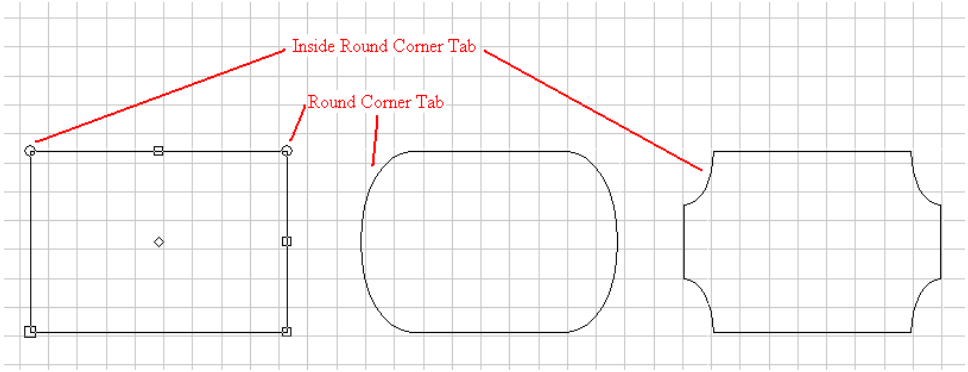
Starting with a simple rectangle or square go to 'Transform' and then choose 'Round Corners'.



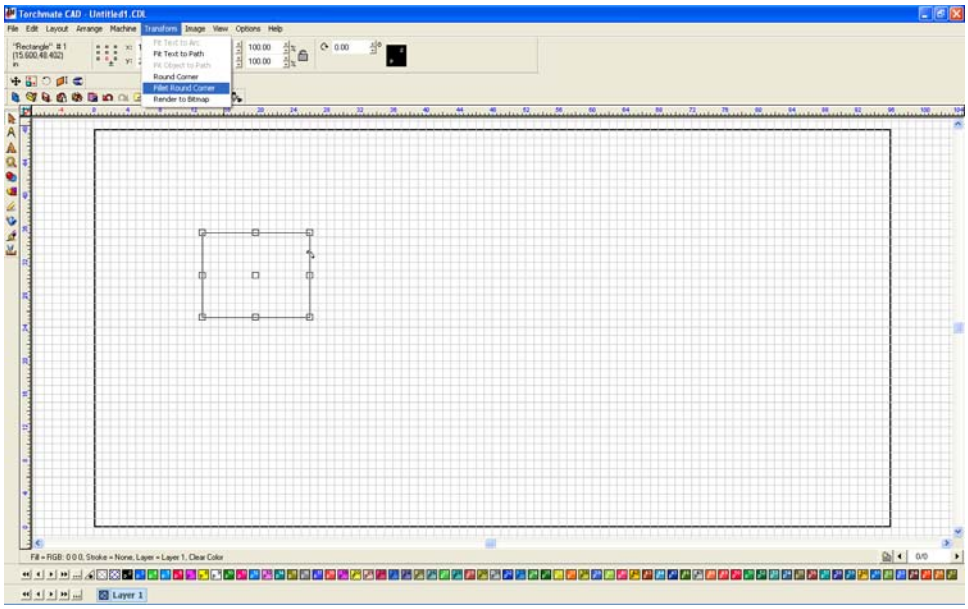
When ‘Round Corners’ is enabled the top menu will have a dialogue box that allows the radius to be specified.



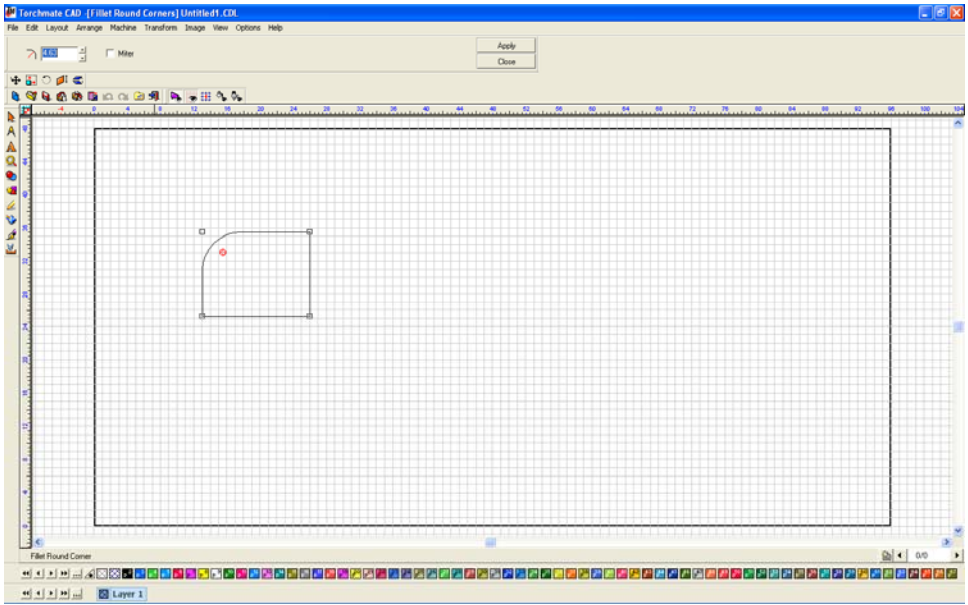
The rectangle tool also has sliding tabs that can create rounded corners. Select either of the tabs and slide to change the radius, there is also an entry box on the top tool bar that allows a specific radius to be set.



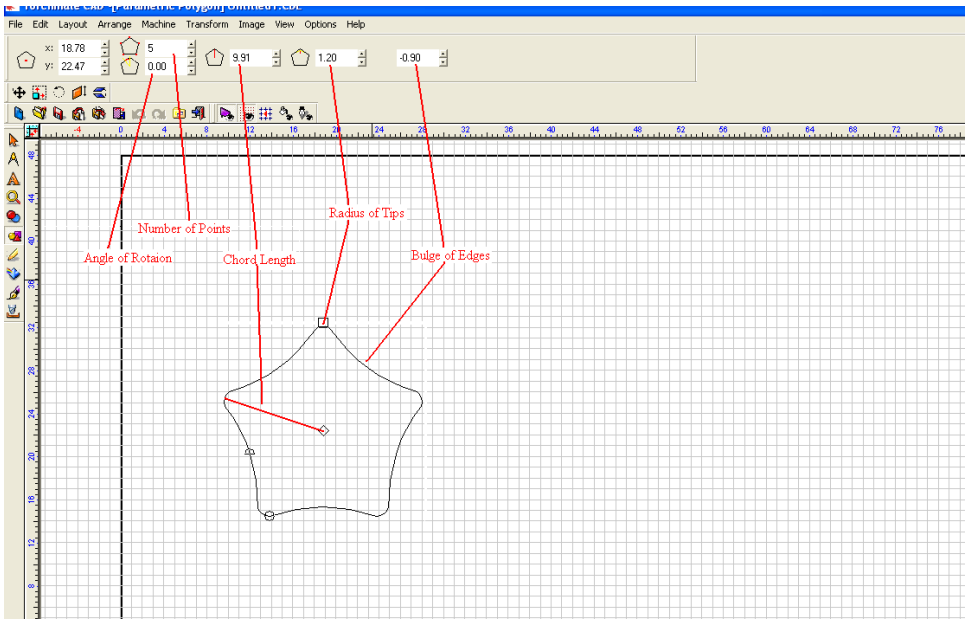
‘Round Corner’ rounds all four corners of a square. In order to round one corner at a time, or to round corners on non-square shapes use the ‘Fillet Round Corner’.



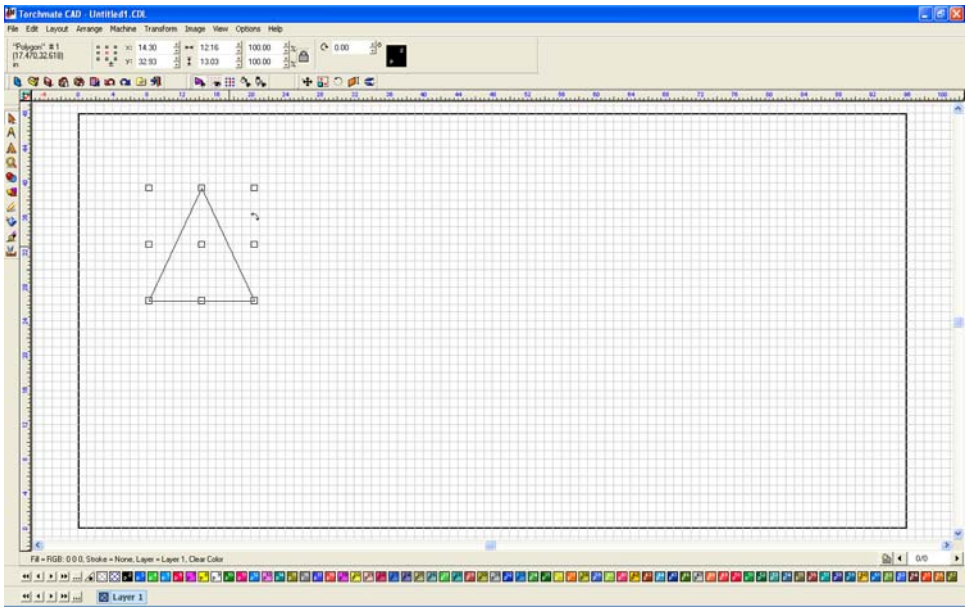
A similar toolbar will appear on the top. In order to specify which corner or angle to be rounded click on the corner, when done click apply then close. The ‘Fillet Round Corner’ tool can be used multiple times to make various radii sizes on different corners.



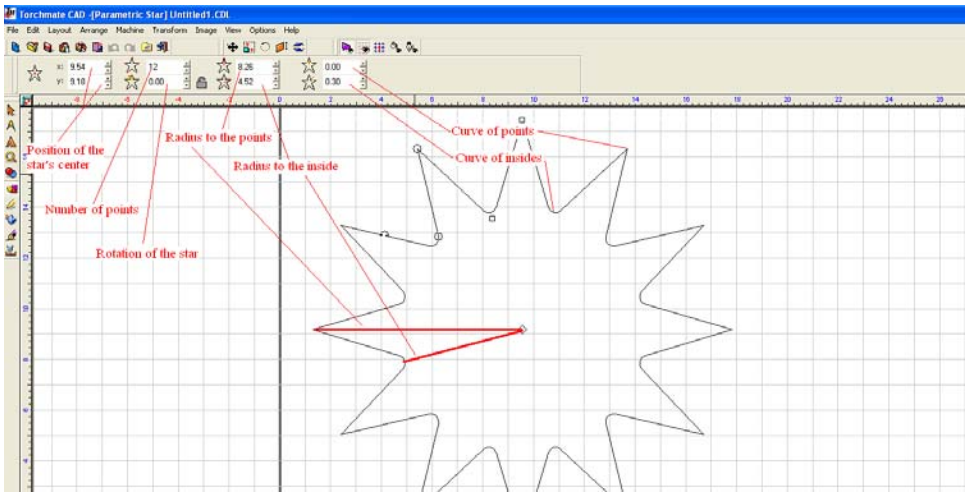
The next tool is the ‘Polygon’ tool. This tool allows the creation of polygons ranging from three to thirty sides. When editing the polygon there are a few fields that can be specified to allow for more advanced shapes, specifically the radius of each tip and the radius of each edge.



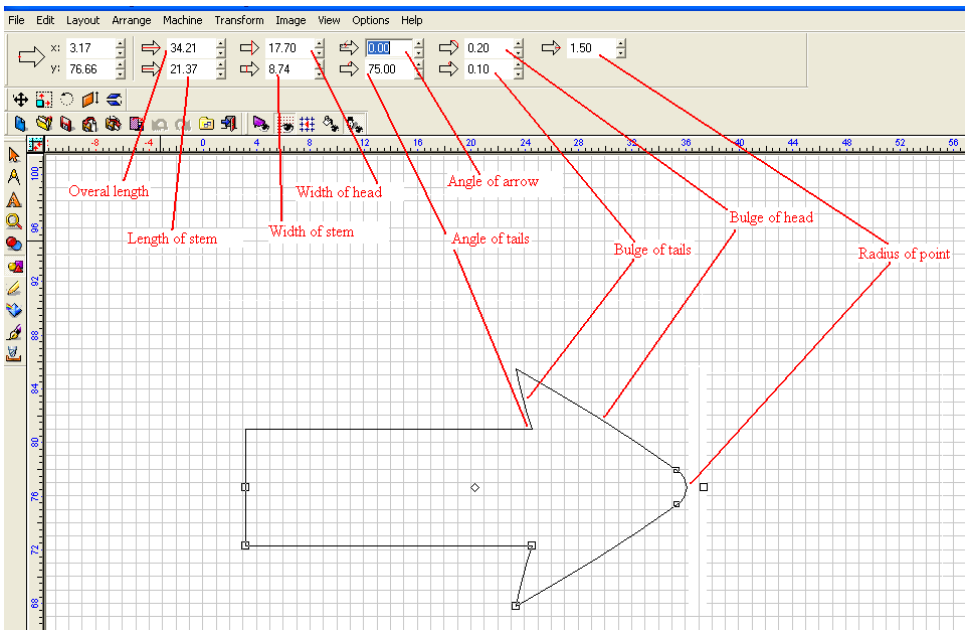
Triangles can be made with the polygon tool, just change the number of sides to three.

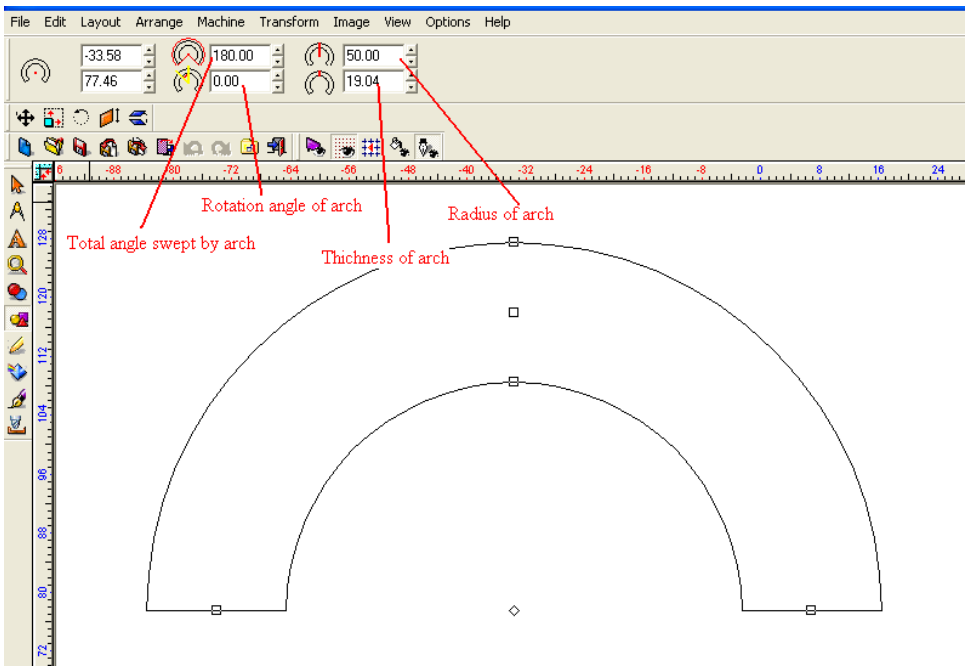


The ‘Star’ tool has similar settings to the ‘Polygon’ tool with a few more added in. A star can be made with three to two hundred points; each point can have a radius point as well as a radius inside.

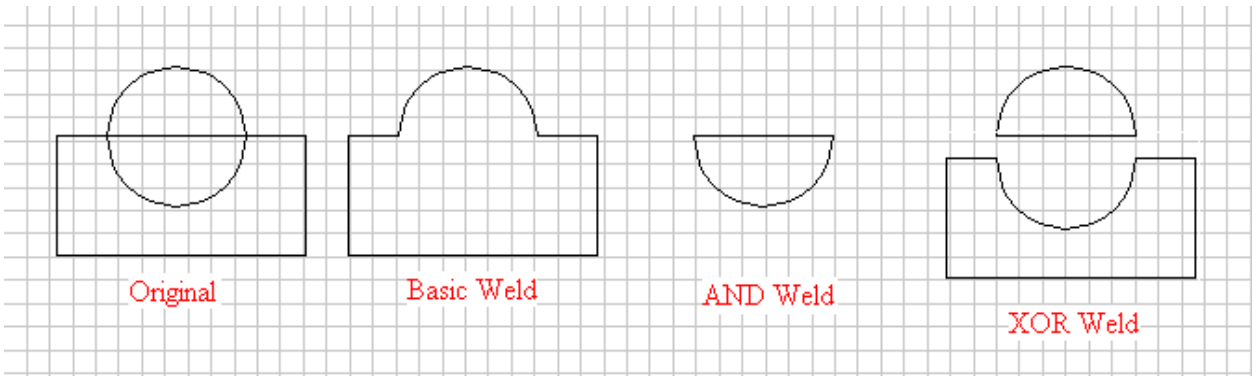


Torchmate CAD can also create arrows and arches. These shapes have just as many features as the other shapes.



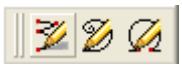


The weld tools included with Torchmate CAD allow the combination of shapes to create cutouts or tabs along with other shapes. To use a welding tool select both objects desired to be welded and then select the appropriate weld. One thing to keep in mind while using the welding tools is that all three operate best when only two objects are selected; weld tools also only work if the two objects overlap each other, even just slightly.

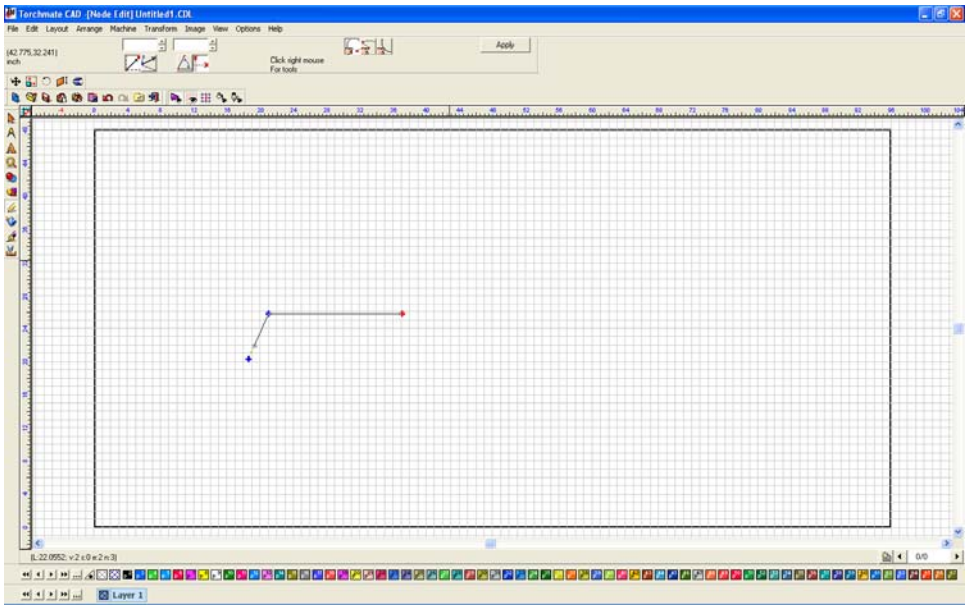


Node Editing:

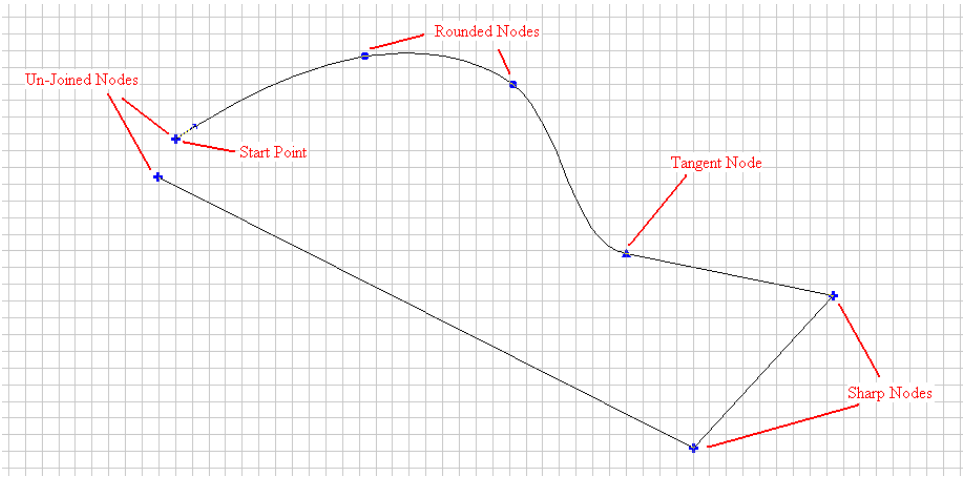
Nodes can be used to create shapes as well. Nodes are defined points that are connected by lines. In Torchmate CAD each node is generally blue and when selected the node turns red. When a node is selected on the top bar there will be two coordinates that can be changed to move the node to the desired point. When a node is deleted the nodes on either side will join together.



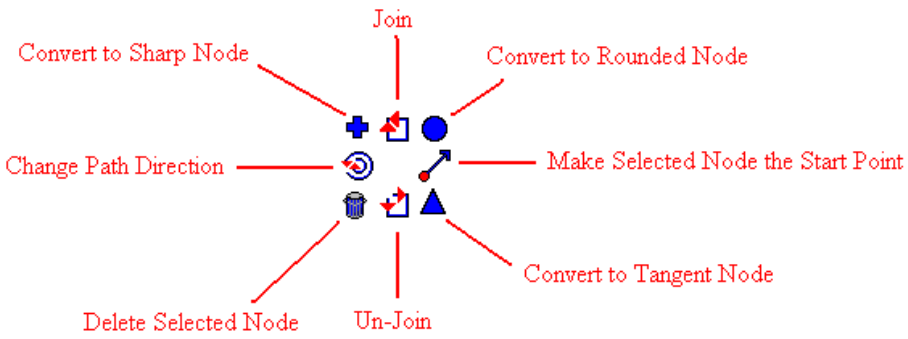
With the node edit tool selected simply click on the work area where the node is to be placed. As each subsequent node is placed the line will extend along the path. To exit out of node editing select the apply button, or if a closed shape is desired double click on the starting point to connect the path fully.



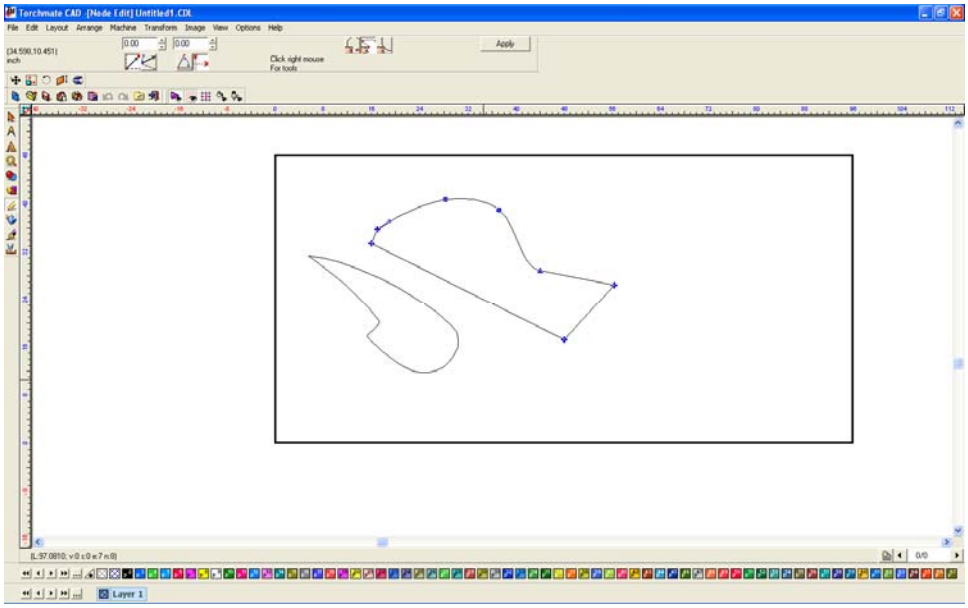
There are many distinctions to be made between nodes. On every shape there will be a single node that has a small arrow originating from it. This node is the 'Start Point' this node is the first in the sequence and the direction of the arrow. If the drawing is not closed there will be two un-joined nodes, these nodes can be joined to other shapes, or joined to each other. There are three types of nodes as well, 'Rounded', 'Tangent', and 'Sharp'. A rounded node defines a curve; this node references its two adjacent nodes for curve information editing any of these nodes will change the dynamics of the curve. A sharp node is used to create the straight lines and angle changes between lines, sharp nodes are the simplest nodes as they basically pin the curve to a point. A tangent node is used to transition between rounded nodes to sharp nodes smoothly.



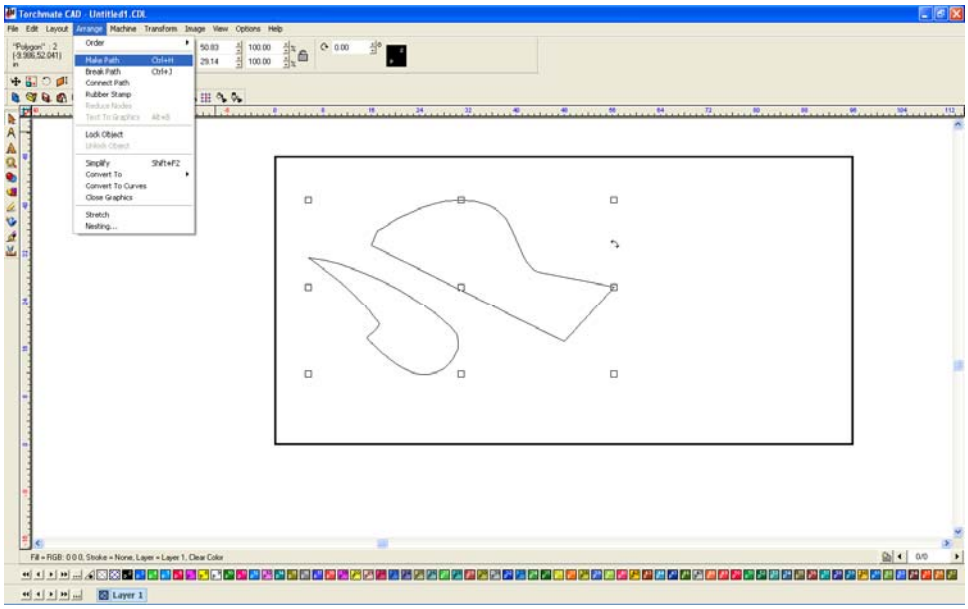
When a node is selected additional features can be accessed by right clicking and holding. Depending on the image type there will be up to eight options. Move the mouse over the option desired and release the right mouse button, this will select the option highlighted.



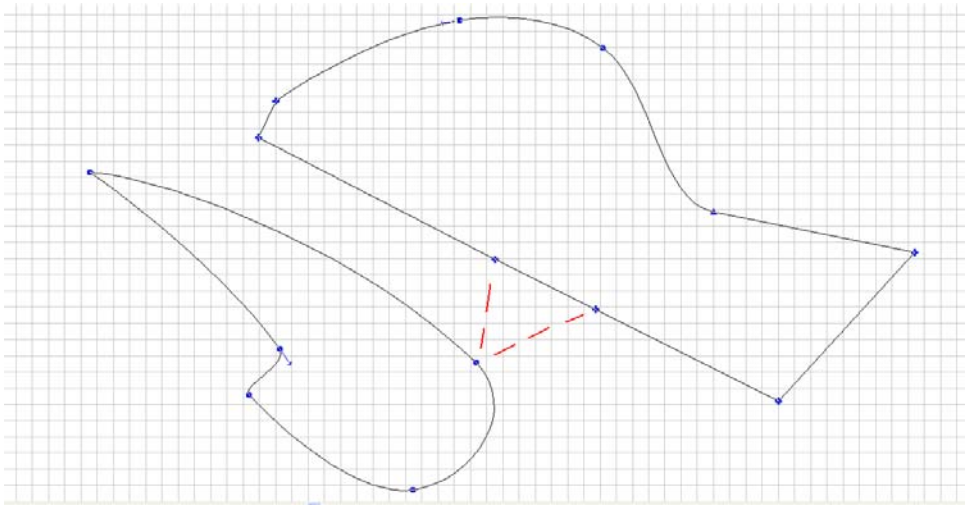
Node editing can be used to join two unrelated objects together. To get into node editing for a shape double click on it. Initially the two objects can be edited but not at the same time.



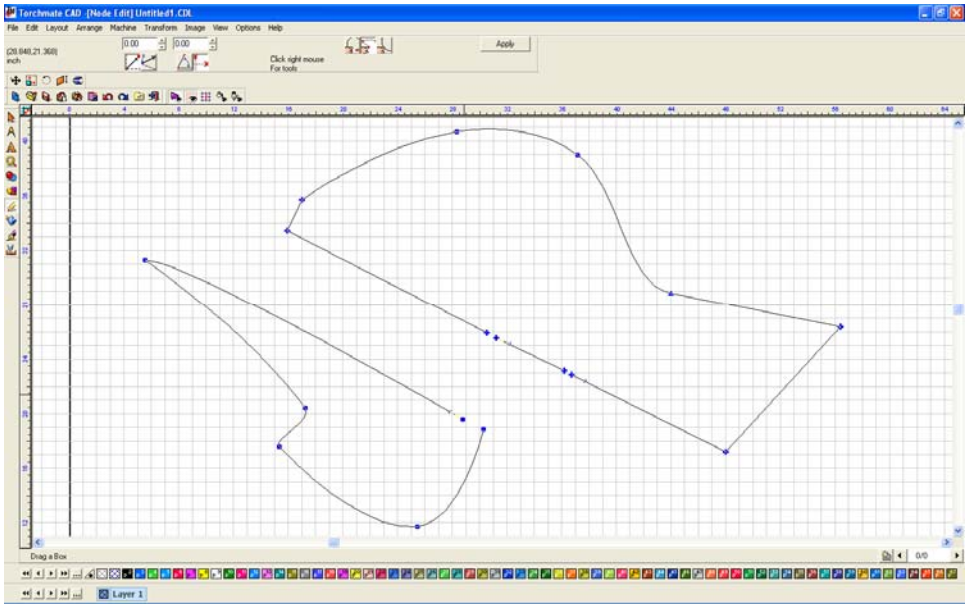
To tell the computer to think of both objects as a single part so that all nodes can be edited at once select both objects and go to 'Arrange' and select 'Make Path'.



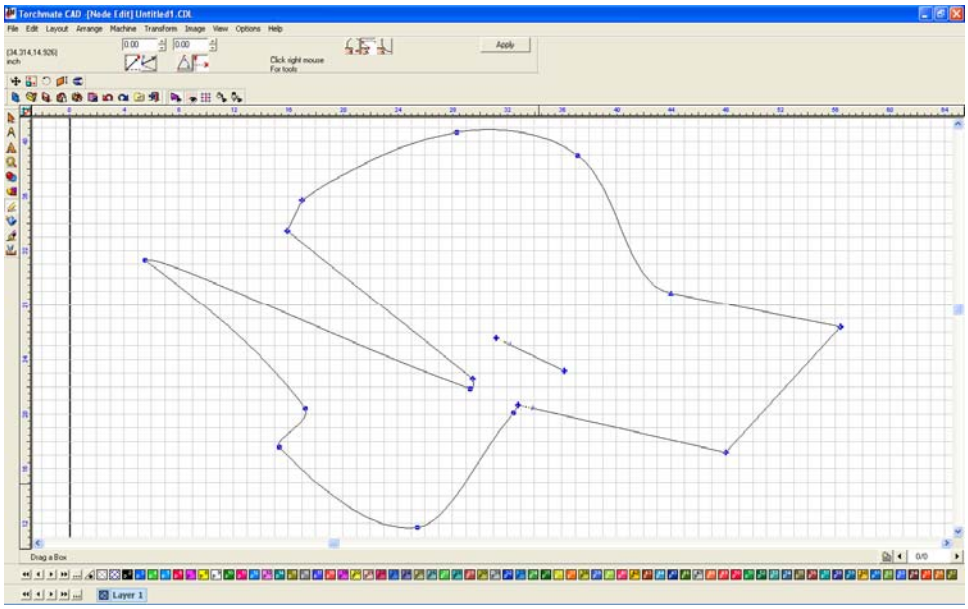
With the paths joined the next step is to edit the nodes to create a bridge between the parts.



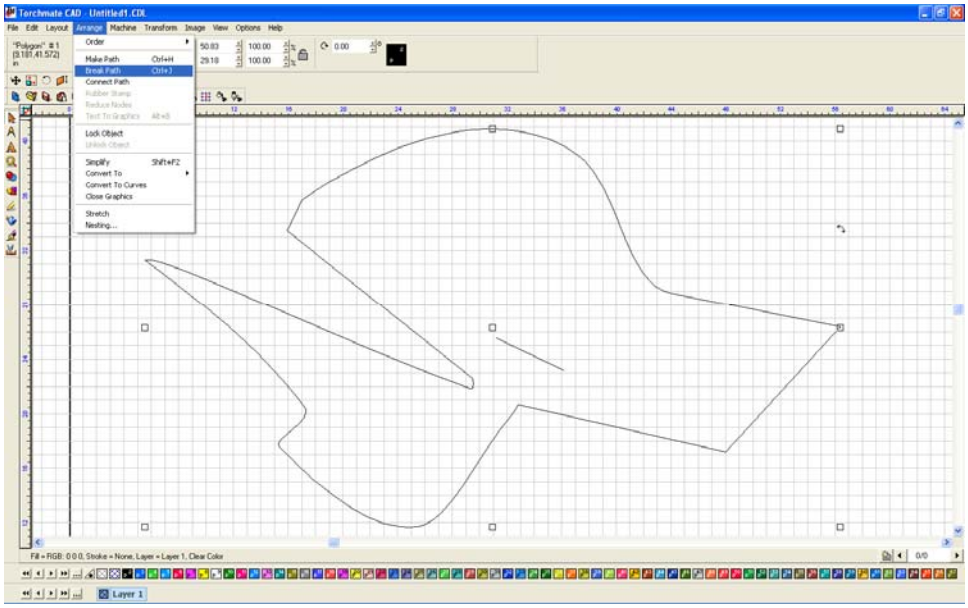
Begin by breaking the nodes apart, select each node individually, right click, and choose ‘Break’.



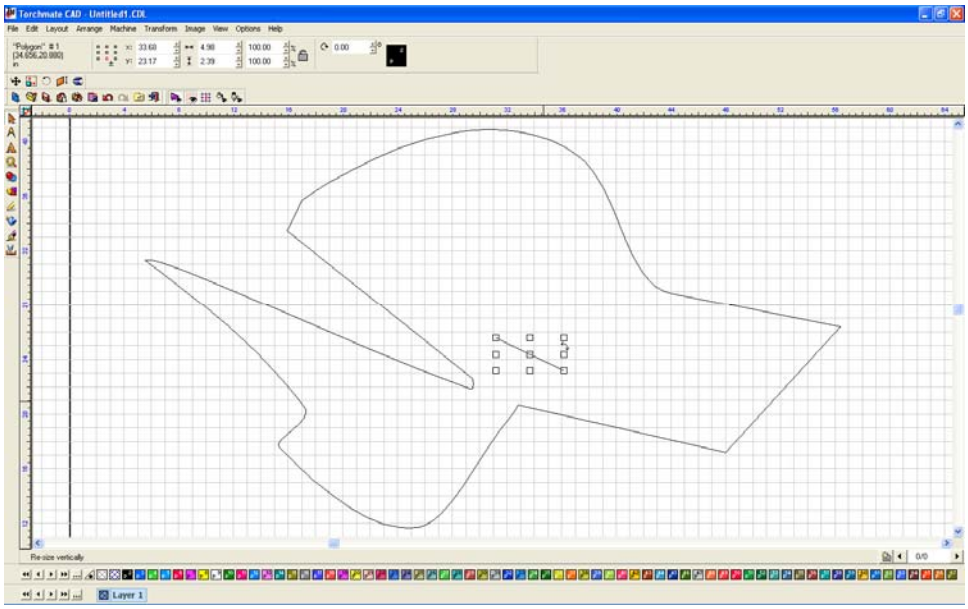
Once the nodes have been broken they can be moved closer to each other. To join the nodes to other broken nodes select both nodes, to do this either draw a selection box around both or hold down the 'shift' key as each node is selected. Once both are selected right click and choose 'Join'. Once both sides of the bridge are joined there is a line left over, this can be thought of as scrap. This scrap piece can be removed a few ways, if it is a fairly simple shape the easiest way is to simply delete the nodes.



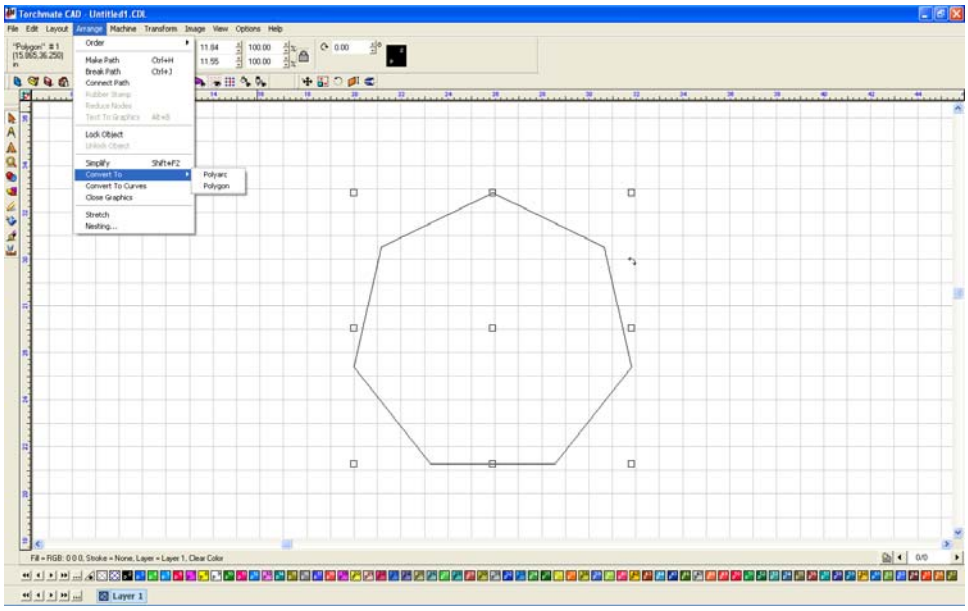
If the shape is more complex the way to delete the scrap is to break the path. Select the part and go to 'Arrange' and 'Break Path'. This separates any un-joined parts into distinct objects.



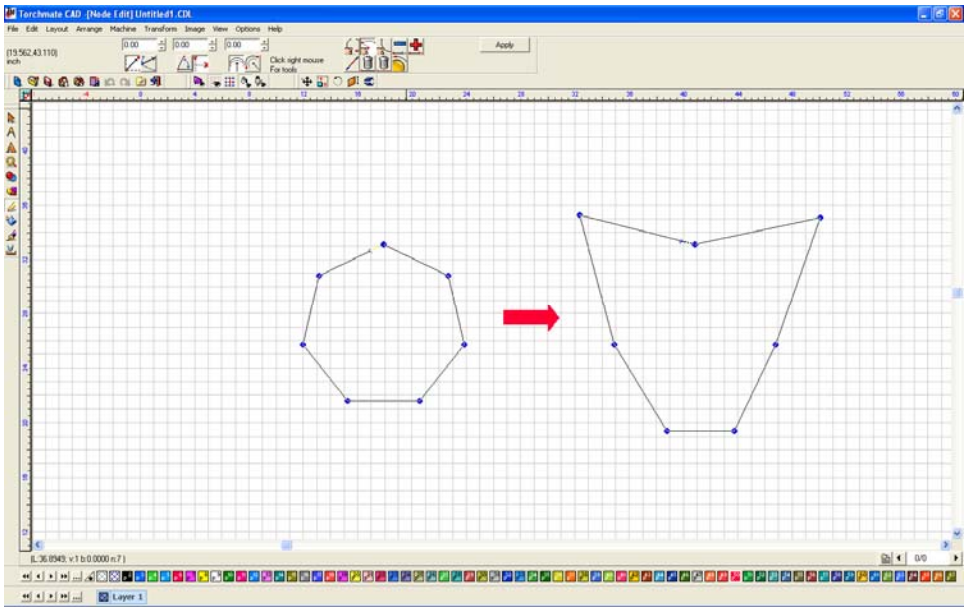
With the objects separated the scrap can be selected and then deleted.



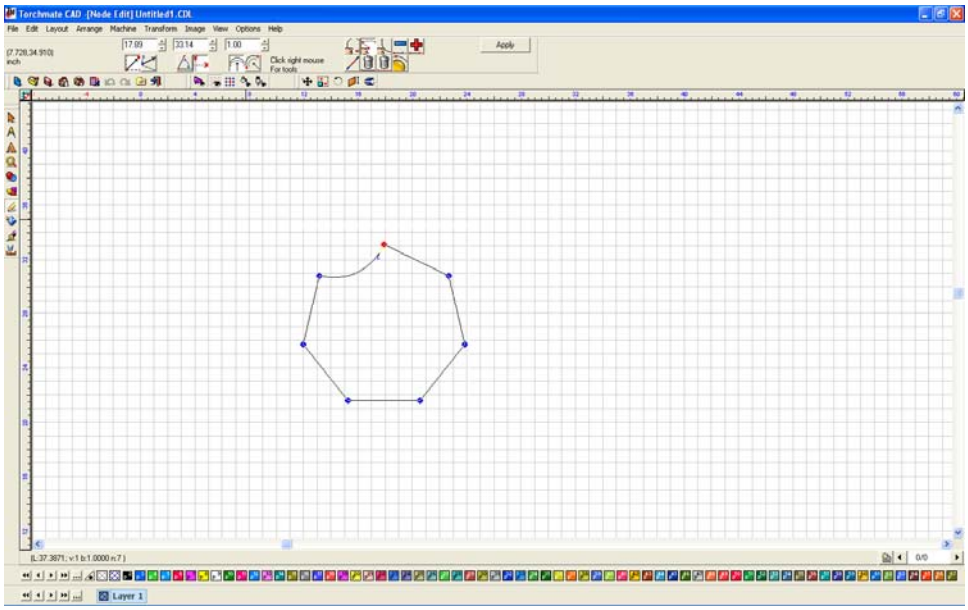
Nodes can also be used to edit existing shapes, even shapes that are created with shape tools. For any shape created with the shape tools there is one step that must be done to be able to node edit. With the part selected go to ‘Arrange’ and ‘Convert To’ select either ‘Polygon’ or ‘Polyarc’. Double clicking on the object will now bring up node editing. The process of converting will not allow the part to be edited further by the shape specific tool bar.



Once converted nodes can now be edited.



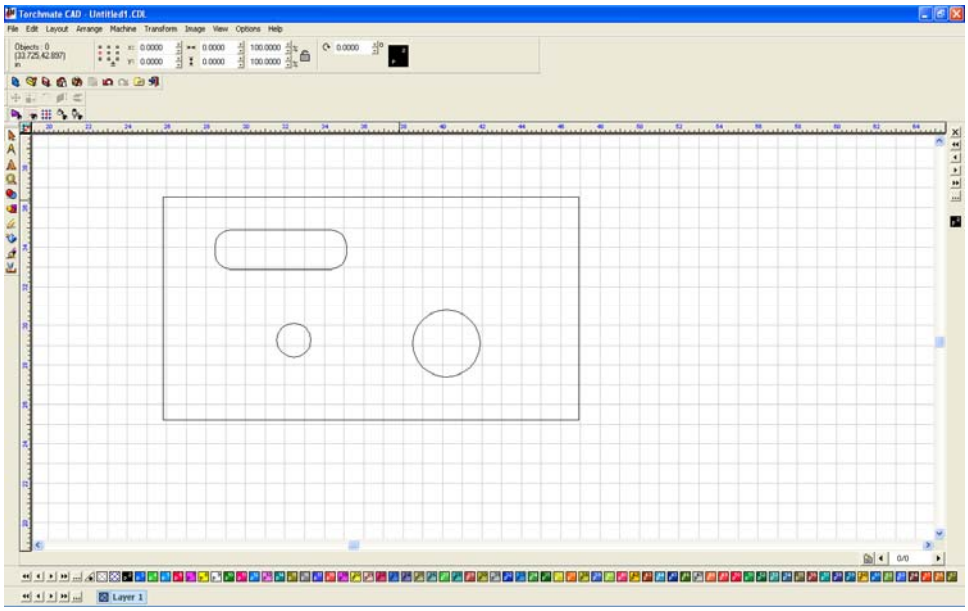
If the object is converted to curves or polyarc the edges can be changed into curves by hovering the mouse over the edge, it will appear as a hand. Clicking and dragging will adjust the curve.



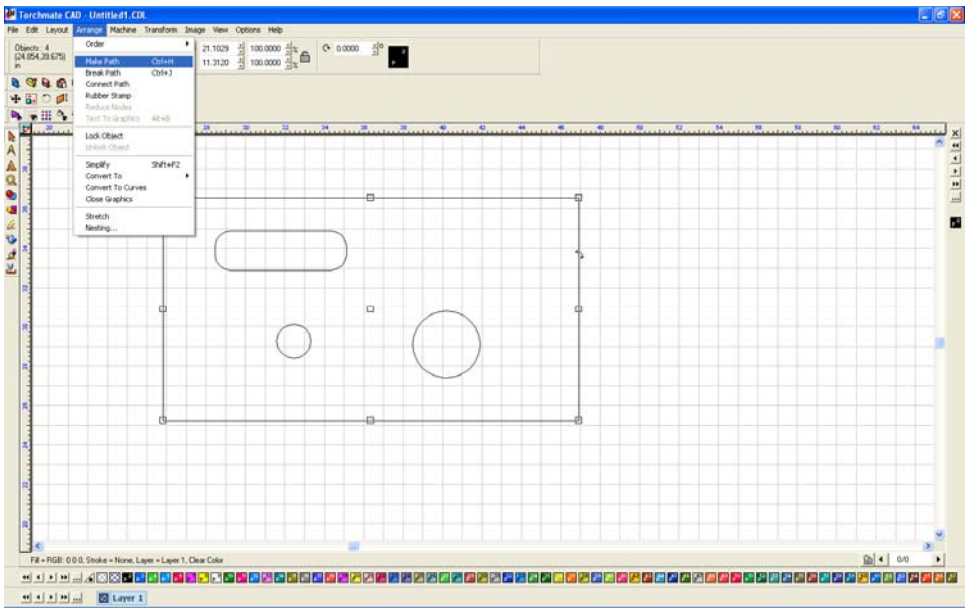
Understanding Paths:

In Torchmate CAD a path can mean a few things mostly it refers to either a tool path or an object. When a path refers to an object the path is essentially the whole part. For simple shapes the path is just one object. Once a part contains holes or cut outs then the path contains multiple shapes. In the Torchmate CAD the path of an object can be broken, connected, or made all path options are accessed from the 'Arrange' menu.

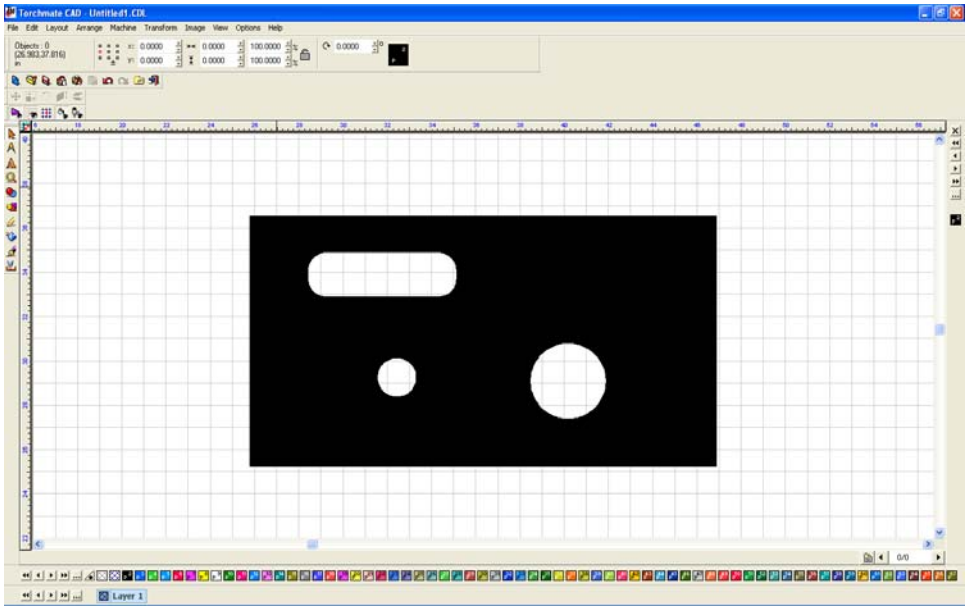
Make path takes multiple objects and tells the CAD to consider them as a single part. Make path is used primarily to create holes in objects. Start with an outline and a few holes.



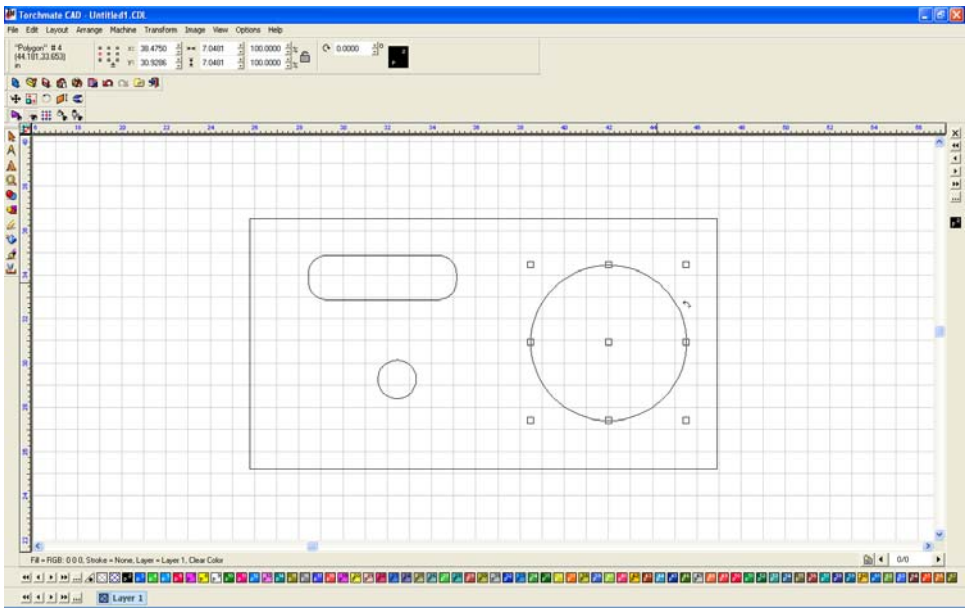
The part when it is just created is four separate objects the objects need to be merged together to one part. Select all objects that compose the part, then go to 'Arrange' and select 'Make Path'.



The objects are now one part and one path. The best way to ensure that this is the case is to go to ‘View’ and select ‘Show Fill’ this displays the object as the CAD interprets it; the filled in parts indicating the actual part.

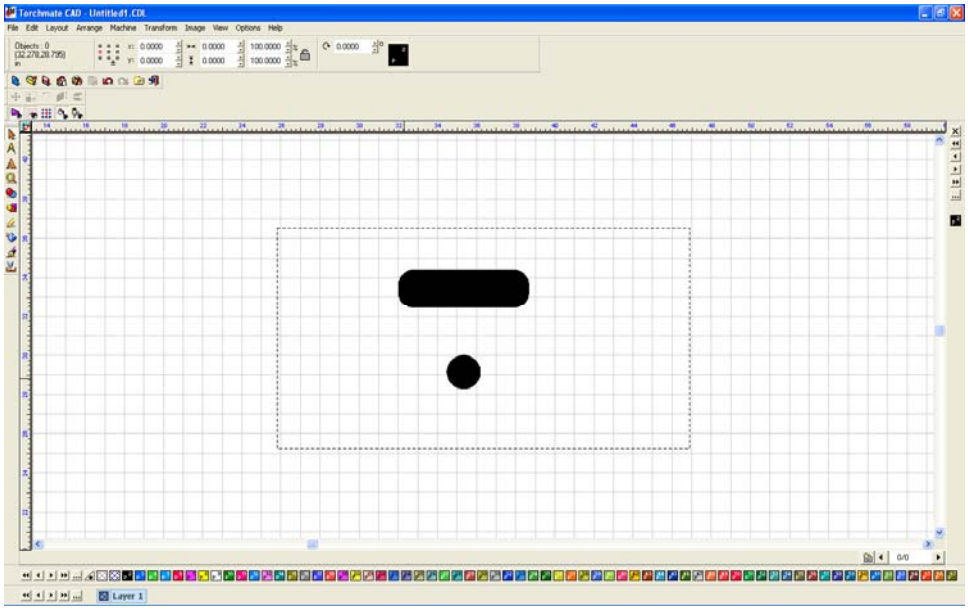


The object is now a single part that can be moved around, all of the nodes can be edited at once as well. To edit a single hole or the outline without changing any parts use break path the separate the objects back to individual elements. Any element can now be changed. Once changed select all parts and ‘Make Path’ again to revert the objects to a single part.

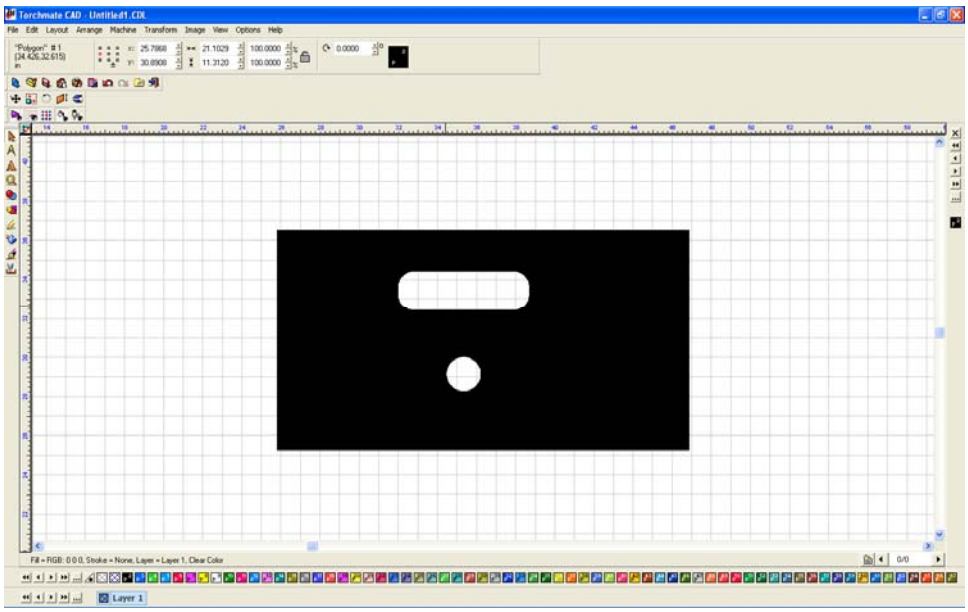


Using ‘make’ and ‘break’ path allows for the editing of single components of a part or the merging of elements to create a part. Show fill is an invaluable tool for keeping track as to what stage a part is in as it will show what the computer is considering a part.

‘Connect Path’ is used to join gaps in a object, these gaps can be very small and will generally occur in objects scanned or imported in to Torchmate CAD. The easiest indication that an object needs to have connect path applied to it is when ‘Show Fill’ is on; any object containing a gap will show up as a dotted line.



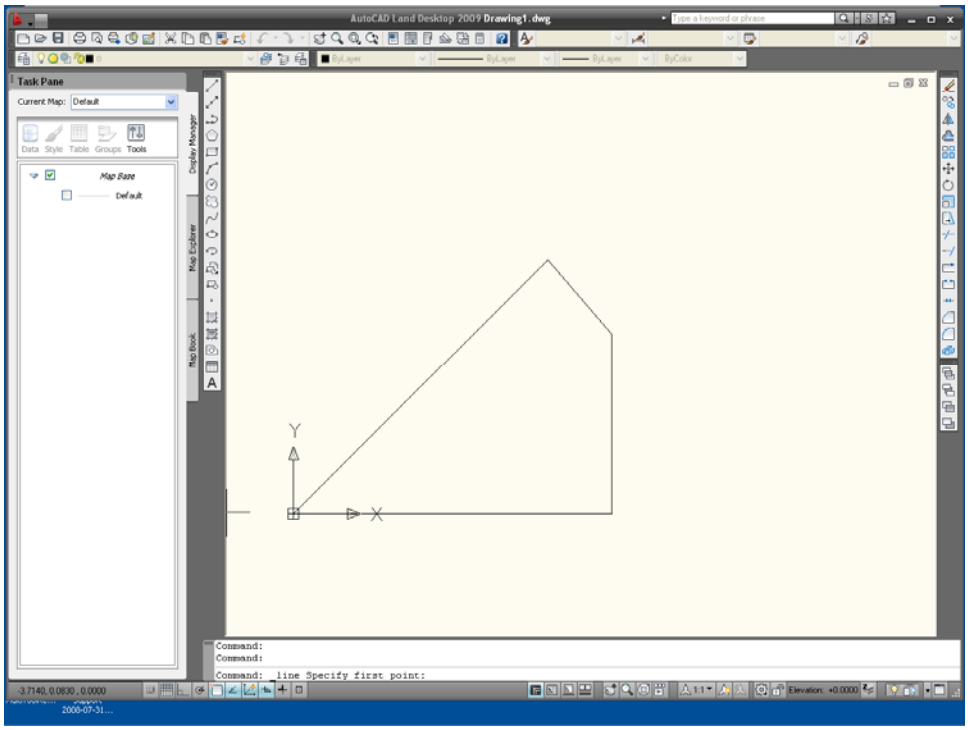
Select the object and choose ‘Connect Path’ from the ‘Arrange’ menu. The CAD will ask for an ‘Allowable Error’ this is the maximum distance it will attempt to join a gap.


Torchmate CAD - Untitled1.CDL
File Edit Layout Arrange Machine Transform Image View Options Help
Layer 1

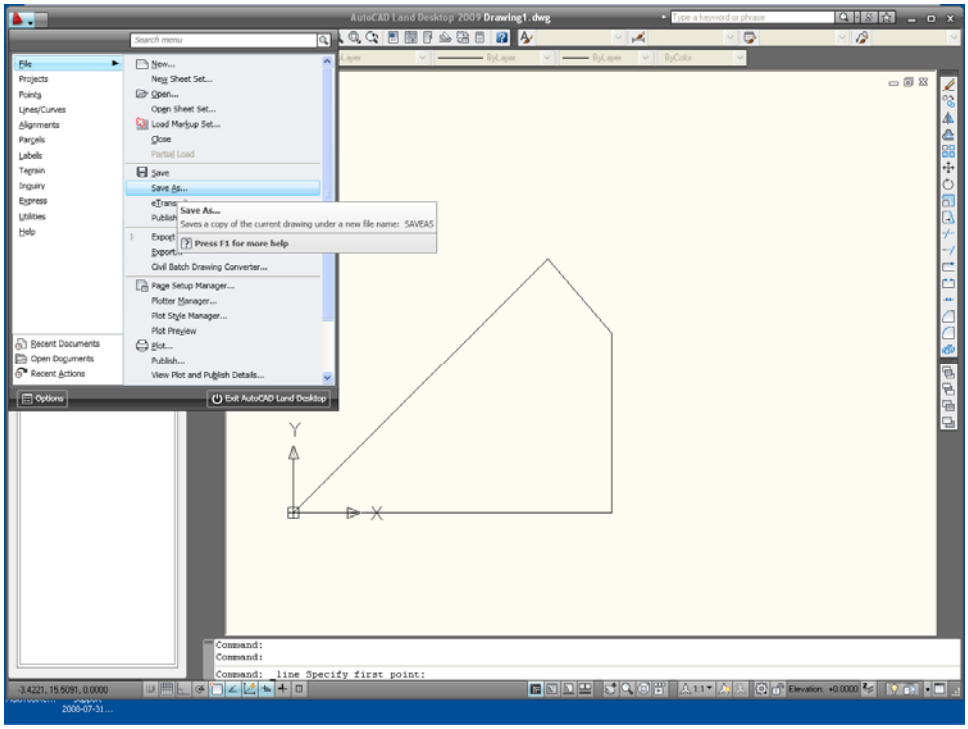
Importing DXF Format Drawings:

The DXF format is a universal exchange format for dimensioned drawings. Torchmate CAD can import DXF files and once in the CAD they can be manipulated and toolpaths can be created.

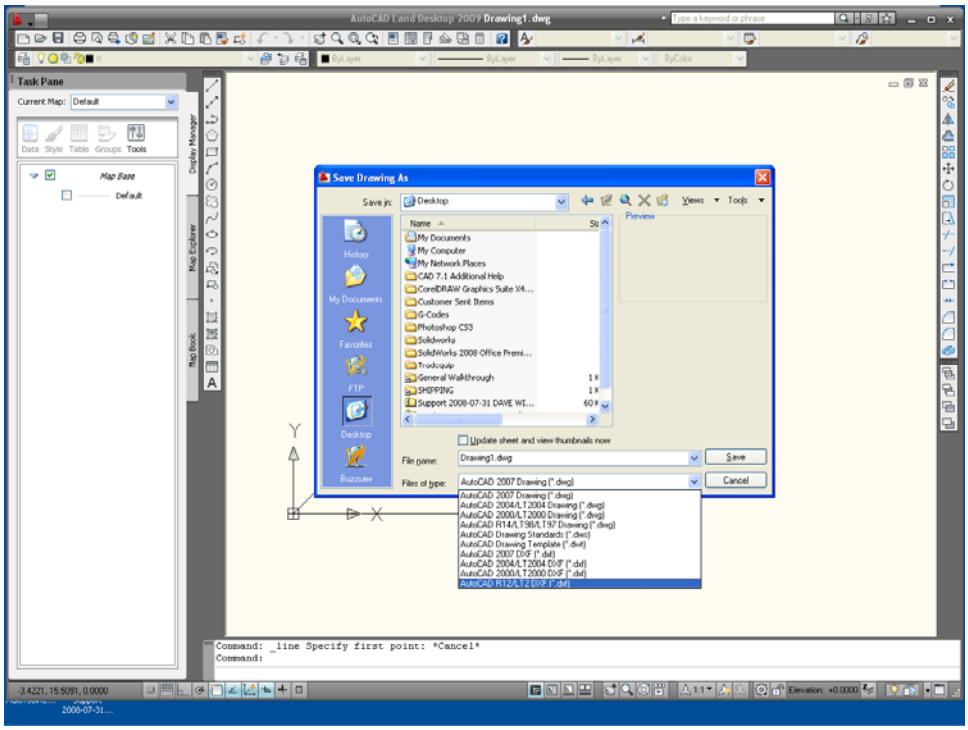
To bring in a DXF drawing the first step is to create one in another program, in this case AutoCAD.



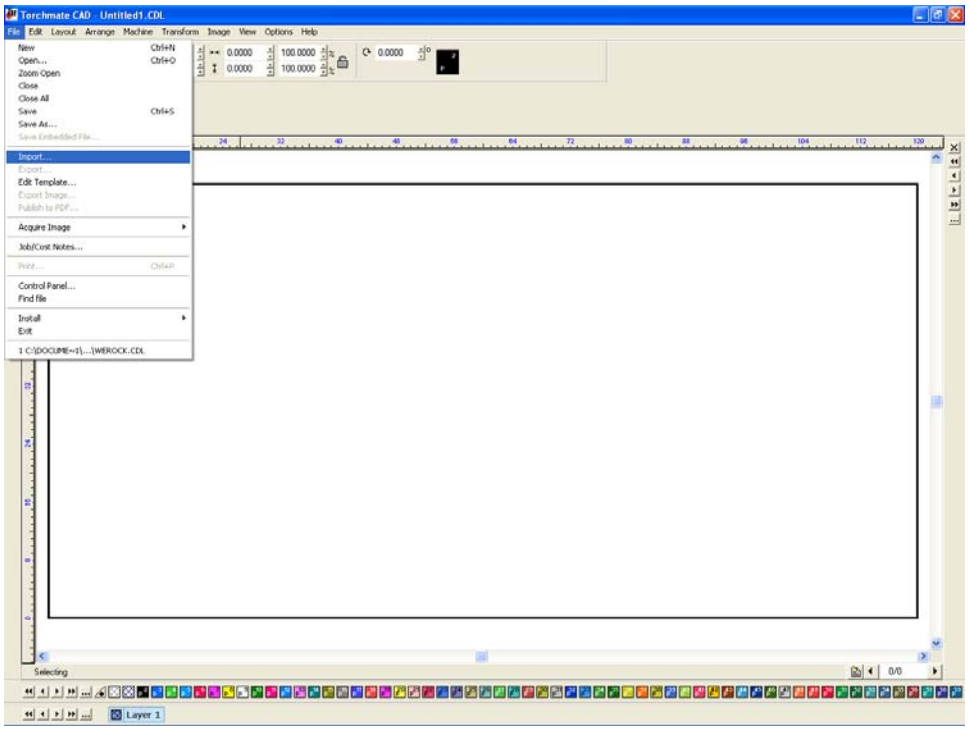
Depending on the program the option to create a DXF may be listed under ‘Save As’ or ‘Export’.



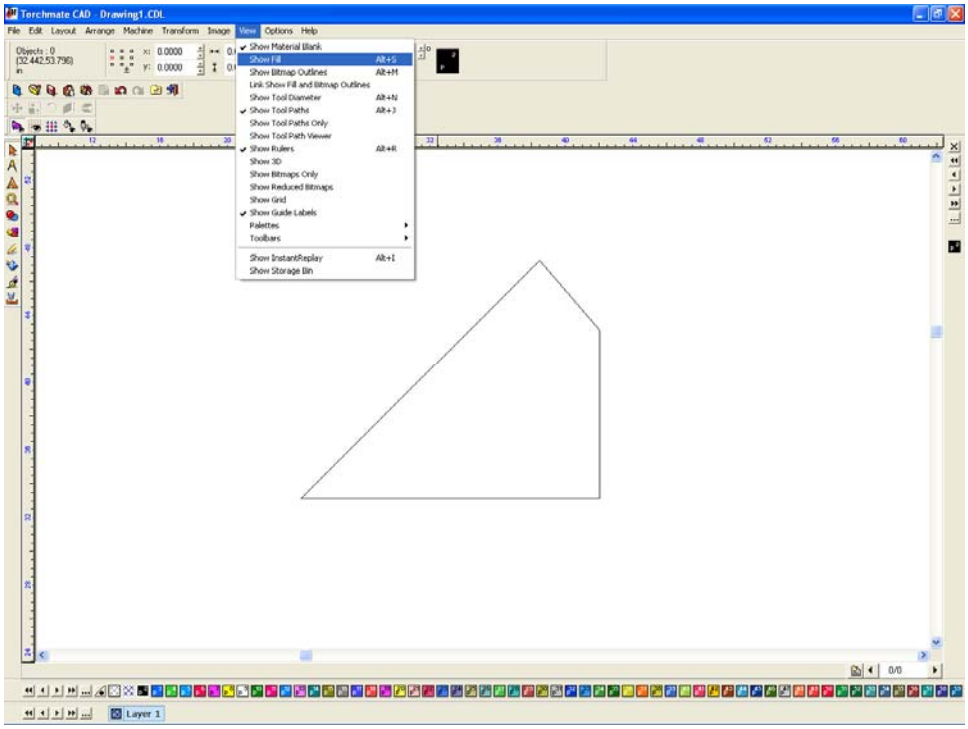
The program being used may have multiple options for which type of DXF to export as some will yield better results than others when brought in to Torchmate CAD.



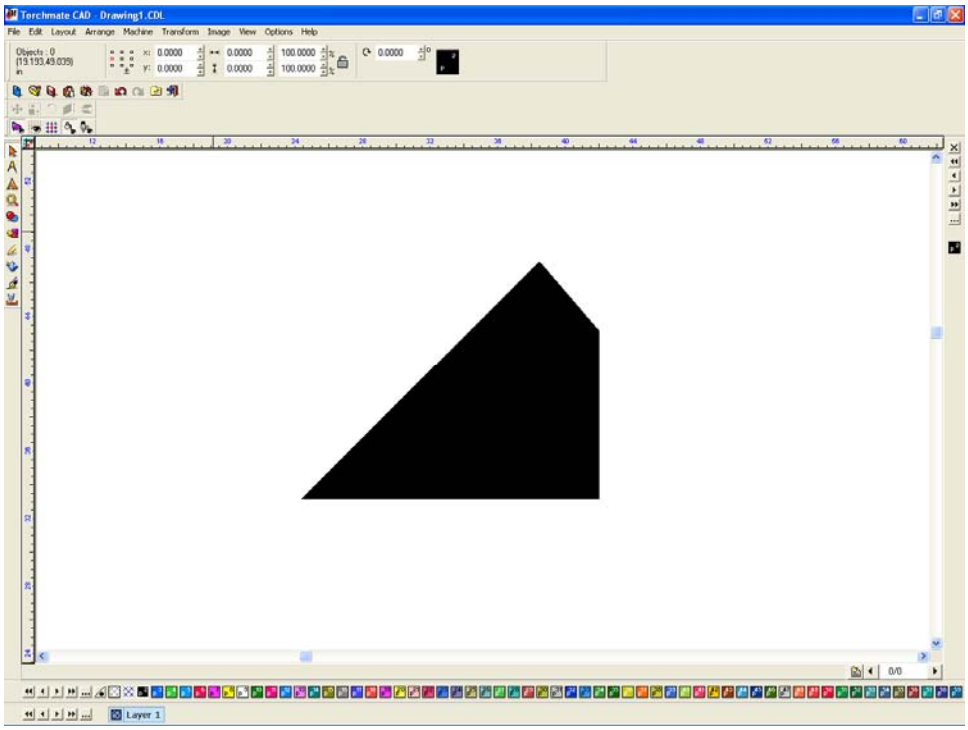
Now that the DXF is saved go into Torchmate CAD, go to ‘File’ and select ‘Import’.



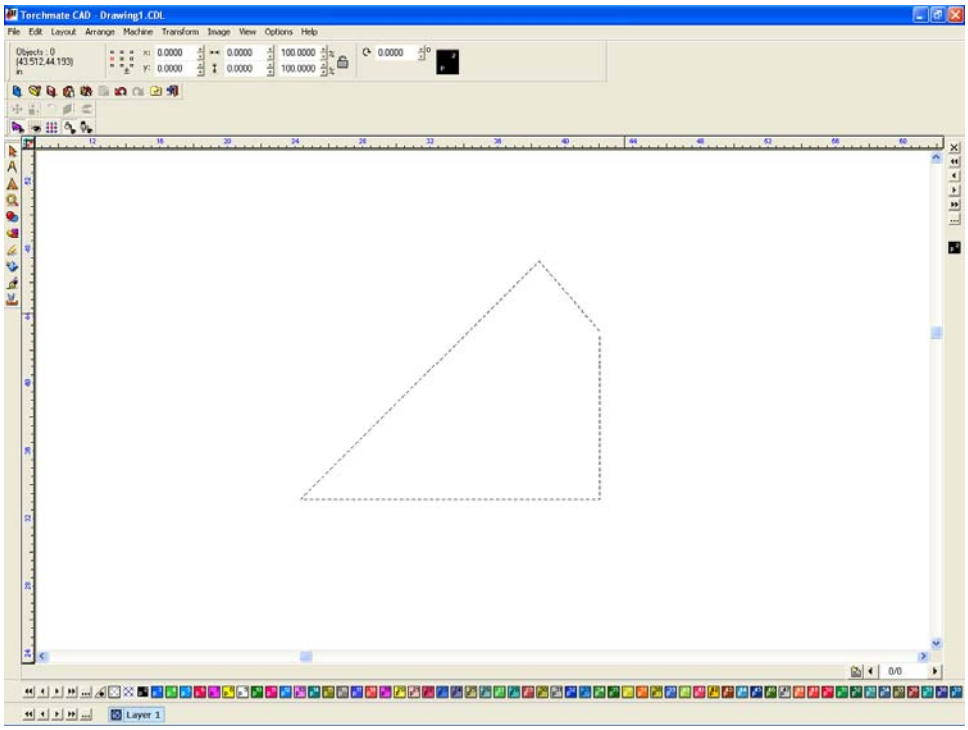
Once the drawing is placed the first step is to ensure that the part has been imported correctly. Go to the ‘View’ and select ‘Show Fill’.



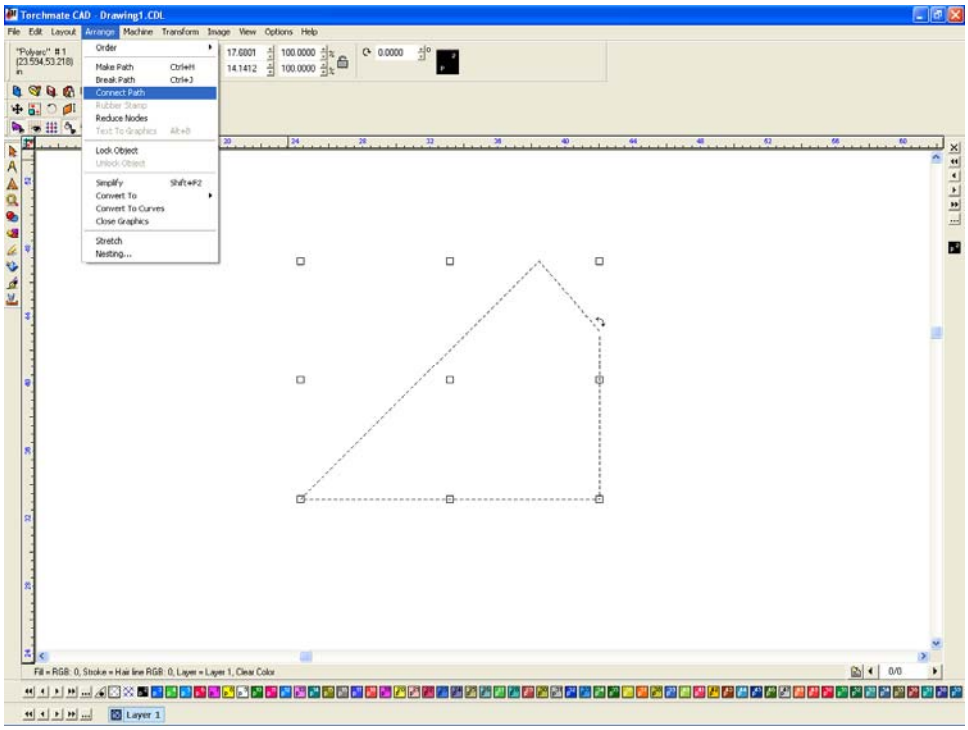
Show fill displays the object as the computer sees it. If the object is filled in solid then it is ready to have a toolpath created.



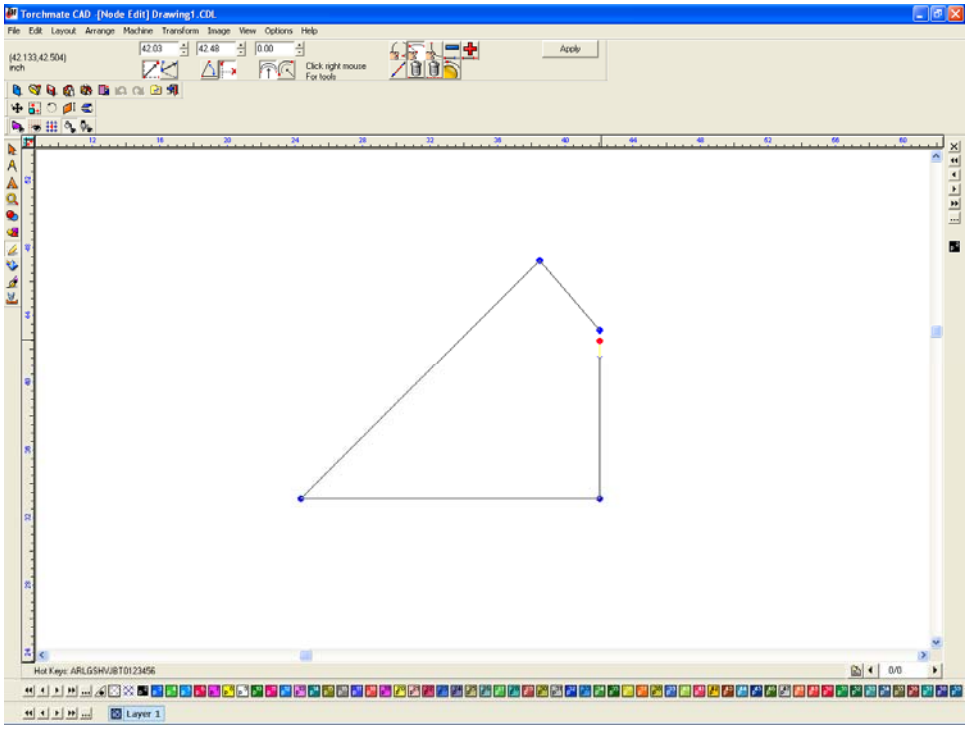
If the object appears with a dotted outline this indicates that there is a gap somewhere.



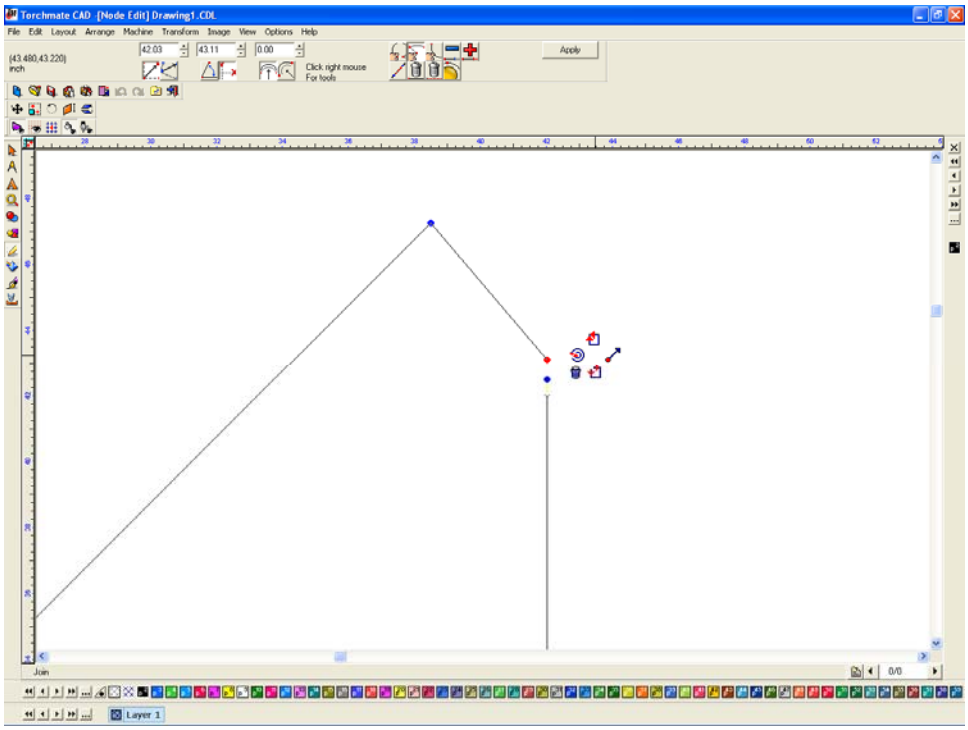
There are a few methods to try to correct this issue. The first method is to select the object go to ‘Arrange’ and select ‘Connect Path’. The CAD will join any gaps within the prescribed gap tolerance.



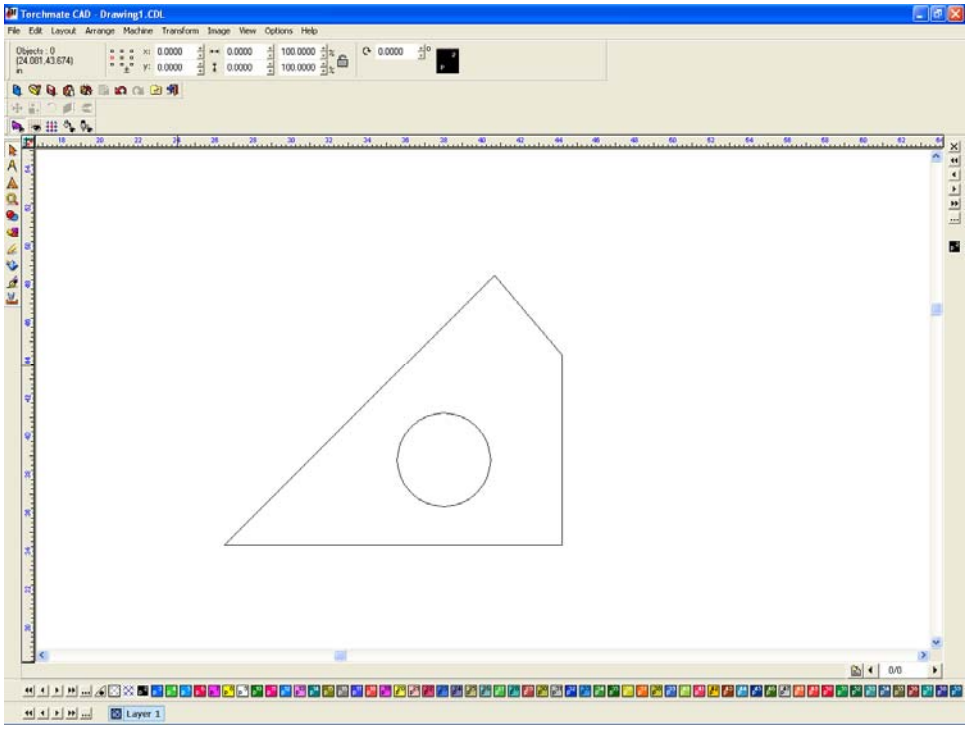
If connect path did not correct the gap the second method is to go into node editing. Double click on the object to enter node editing mode.



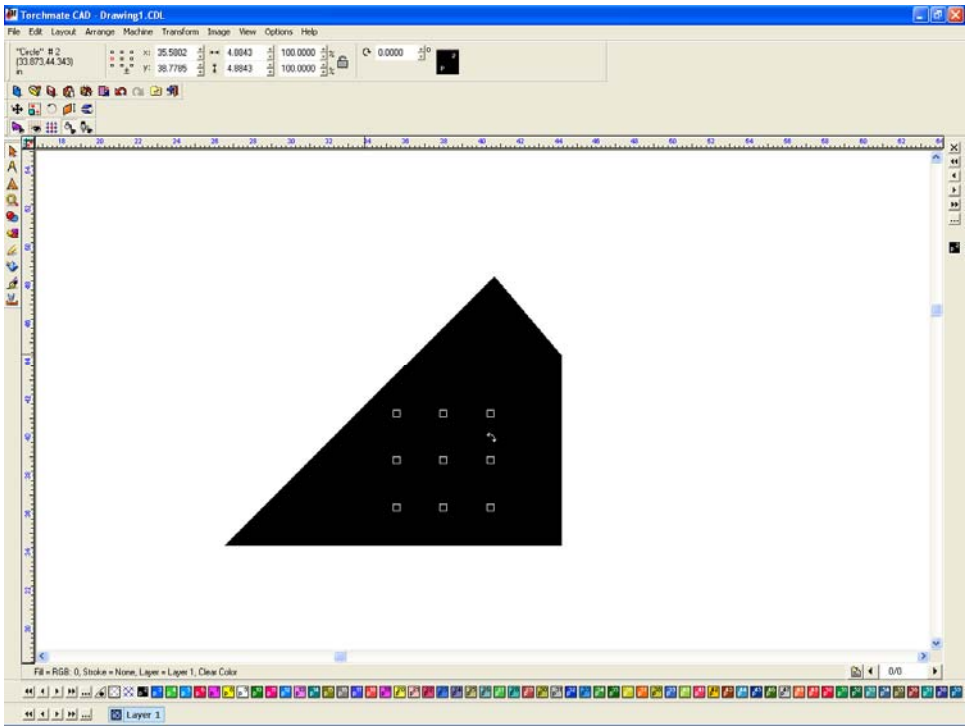
Select any node and it will turn red. If the ‘Tab’ key is held down the CAD will cycle through the nodes selecting each one in sequence. If there is a gap the cycle will stop and pressing ‘Tab’ will do nothing. At any place there is a gap select the node, hold down right-click, this brings up the node options. The top middle option is join this will join the node with a nearby un-joined node. Verify with ‘Show Fill’ that the part corrected.



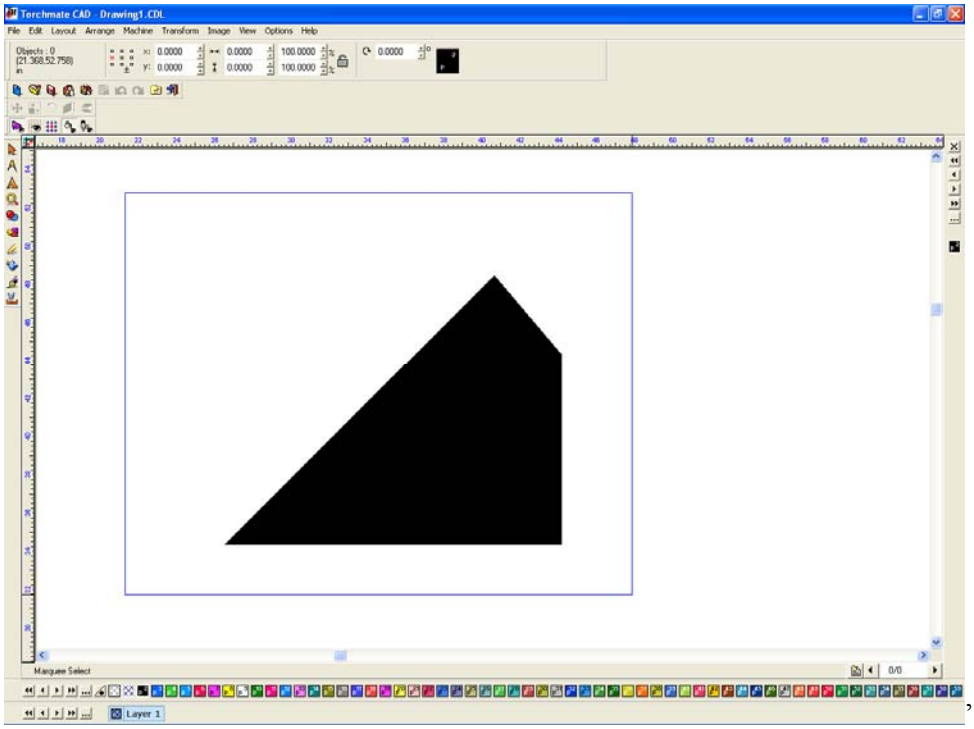
Another issue that can arise when importing a DXF occurs when the drawing has inside and outside parts.



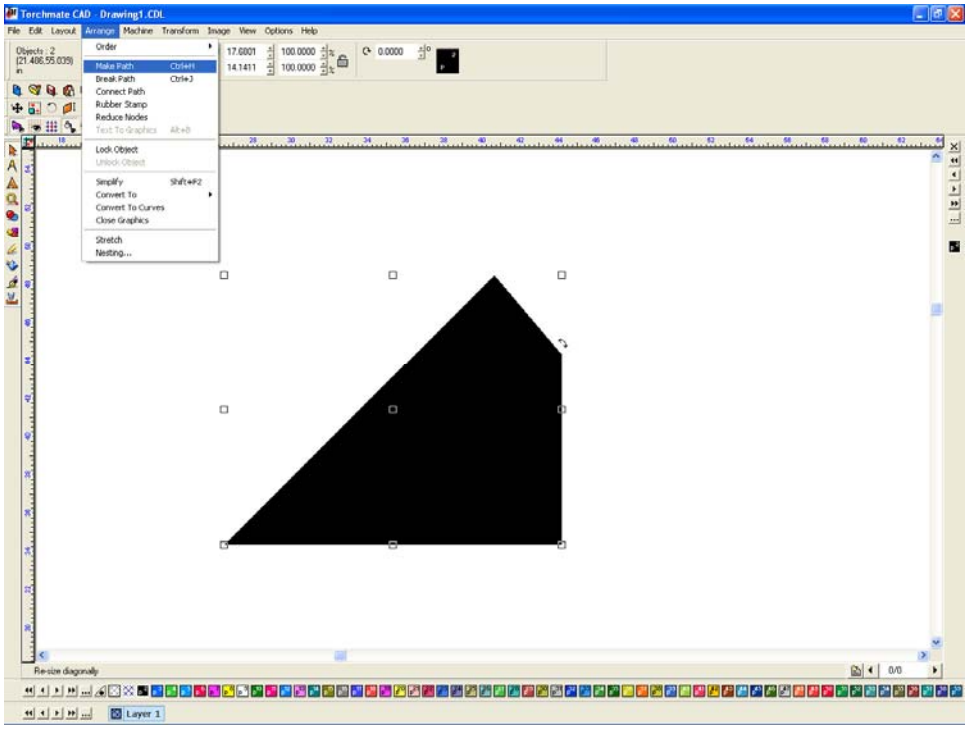
In this case the fill displays each object as solid, the CAD simply does not associate the hole with the rest of the part.



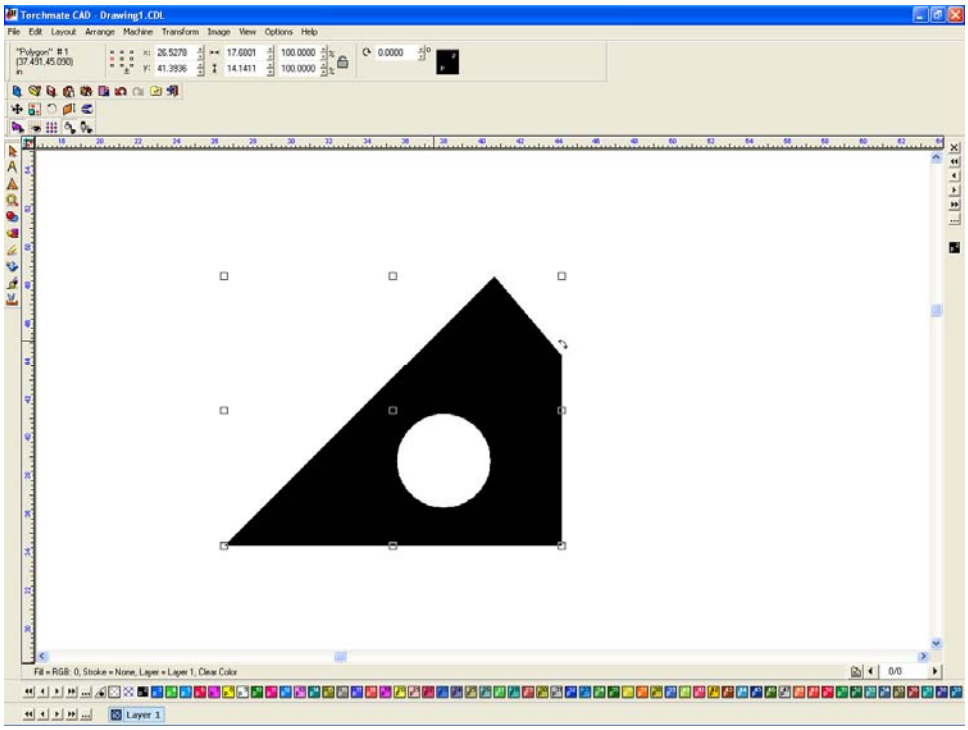
To correct this first select all of the components of a single part.



Go to ‘Arrange’ and choose ‘Connect Path’ this will tell the CAD to think of every object selected as one single part.



Once the part is corrected ‘Show Fill’ will display it correctly.

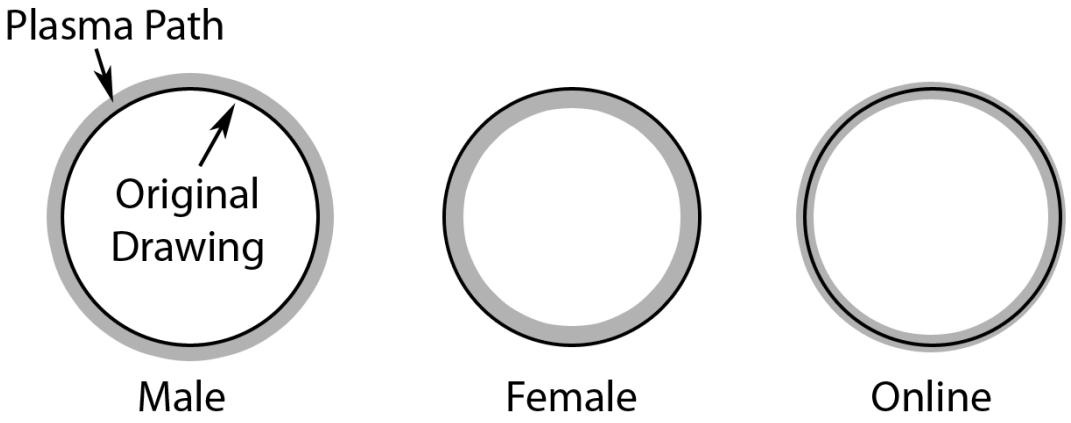


When importing DXF files keep in mind that some parts may have no issues while others can have several, it is always a good idea that before proceeding to cut or edit the part ensure that it is correctly imported.

Frequently asked questions about tool paths.

What is the difference between a Male, Female, and Online tool path?

Tool paths come in three varieties male, female, and online. The difference has to do with how the tool paths are offset from the original drawing. When cutting with plasma there is a certain amount of material eaten away this is called the kerf. In order to maintain the dimensions of a drawing the tool path needs to be offset from the drawing. An outside cut will use a male tool path and an inside cut will use a female tool path. An online tool path does not offset for any kerf and simply created a tool path along the center line of the drawing. Lead ins and lead outs are also created based on these offsets.



I have a part with a hole cut in it, what type of tool path do I use?

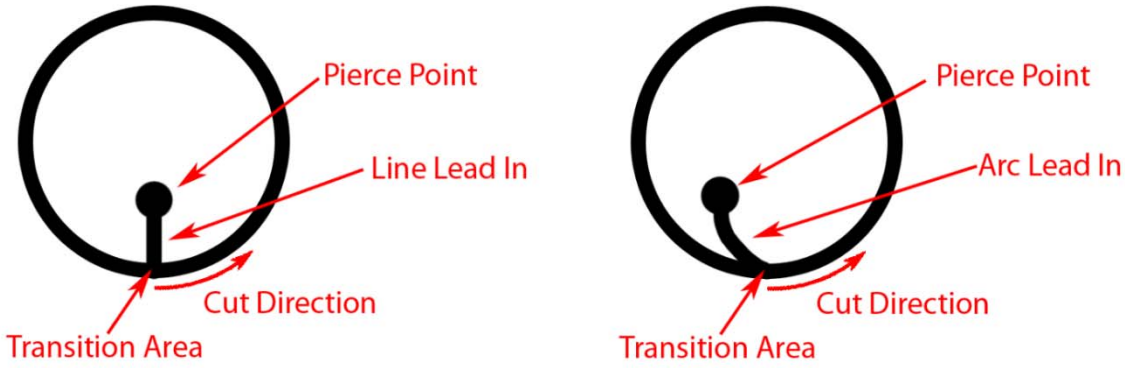
When a tool path is made in Torchmate CAD the type of tool path selected will determine how the outside cut is made, any inside cuts will be made opposite of the specified type. For instance if a plate with a hole in it has a tool path created select a male tool path, this will make the inside holes female automatically, all the critical dimensions will then be maintained.

Lead ins? Lead outs? Do I need both, some, or none?

For plasma cutting it is generally a good idea to create a lead in. What a lead in does is move the pierce point of a cut away from the finished edge and into an area of scrap. This prevents a dimple from being created on the part where the pierce occurred. In Torchmate CAD when a male or female tool path is made the lead in will be created to coincide with the offset direction. With plasma lead outs are generally not used especially if a height control is being used. If a lead out were used the part would finish cutting and then go to the lead out, the scrap piece may shift or fall out the arc will attempt to go somewhere this will result in the torch either diving down to follow the falling slug or the arc jumping to the part edge causing a divot. Oxy fuel torches can use a lead out to do common line cutting or in general to ensure a full sever.

Arc or line lead ins which are better? What is the difference?

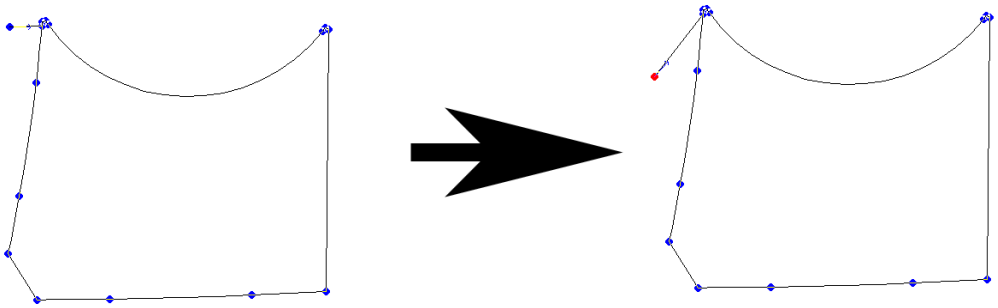
In most cases the difference between an arc lead in and a line lead in comes down to personal preference. The area where one would be preferred to another is when cutting inside holes.



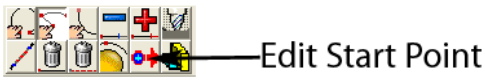
For a well calibrated table with minimal backlash a line lead in should be used, this will result in a cleaner transition area. The issue with an arc lead in occurs when the plasma's path comes around to finish the cut; it will encounter an area that has been partially cut on, and this can result in a divot or excess slag forming in the transition area. In outside cuts or non-circle inside cuts the lead in type generally will make no difference to the final quality.

Can I adjust my lead in points after I have created a tool path?

In Torchmate CAD the lead in length can be adjusted, this is similar to node editing a regular object. After a tool path is created double click on the tool path. The tool path will have blue points along it that indicate direction changes. The lead in will have a node at its end that can be selected and moved to a desired position.

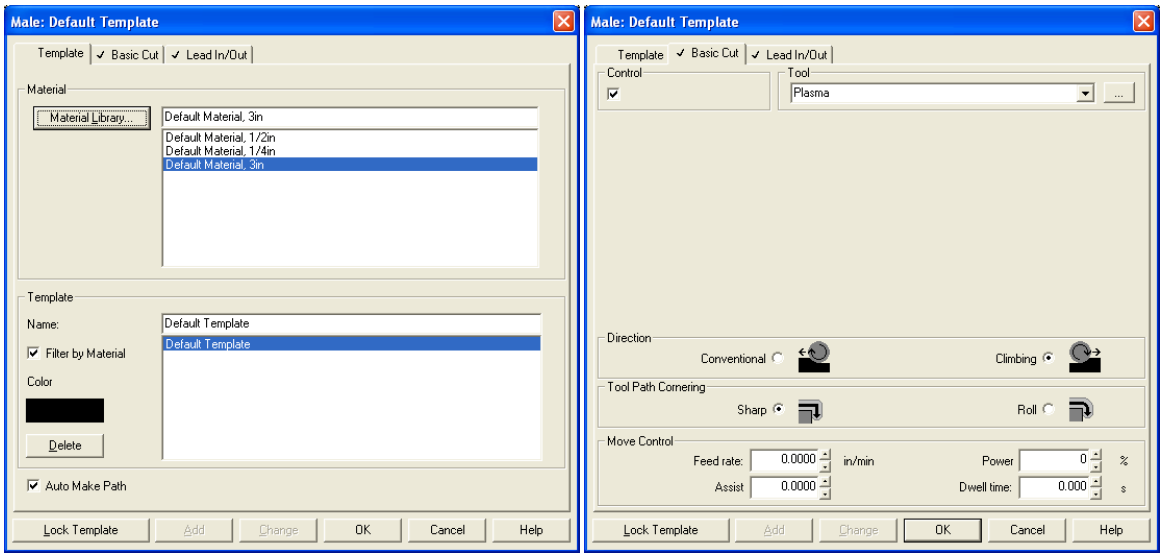


The point where the lead in connects to the part can also be adjusted after the tool path is made. When the tool path is selected and in node editing mode (by double clicking on the tool path) the top tool bar will have twelve small boxes, one of them adjusts the start point. Simply select the start point tool and click on the part where the new start point should be.



When I create a tool path what do all of the options do in the menu.

When creating a tool path there are three tabs that are accessible. The first tab is called 'Template' this tab can be used but will not affect the tool paths if it is never used.



If this tab is used what it stores is the information from the other two tabs as a material. For instance the settings can be set for one thickness, once the settings are made selecting change on the bottom row of buttons updates the selected material so that in the future the settings load with the specific material. Again if nothing is ever changed on this page the tool paths will never be affected.

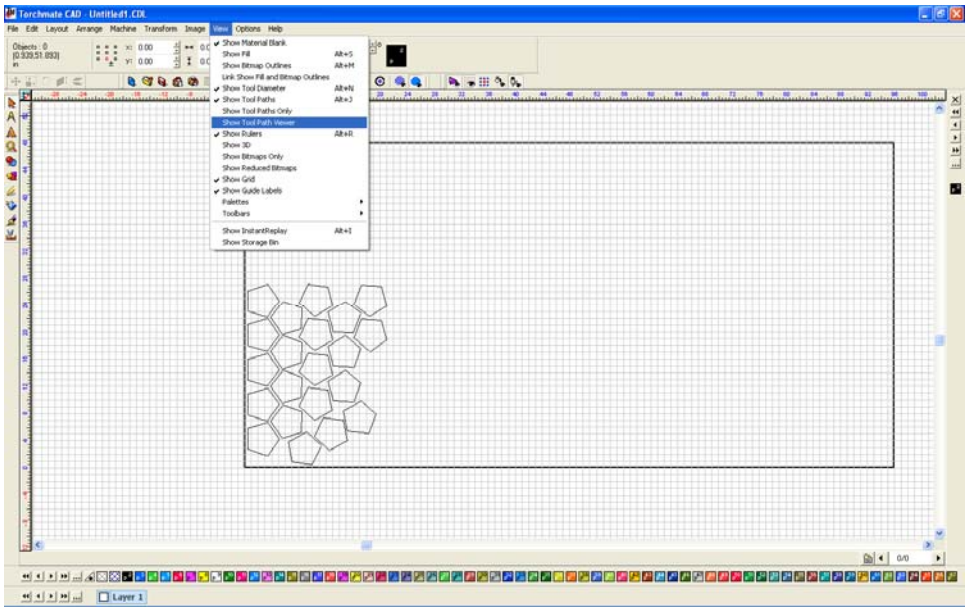
The second tab is called 'Basic Cut' this tab controls the tool being used as well as the shape of the cut. The tool pull down menu allows any tool from the tool library to be selected. The tool is what defines the offset to be used. 'Direction' specifies the motion of the tool around the tool path. 'Conventional' will create cuts so that an outside cut travels counter-clockwise and inside cuts will be clockwise, 'Climbing' will create the opposite direction tool paths. For plasma cutting a 'Climbing' cut should be used, for routing generally a 'Conventional' cut would be used but this can vary based on the bit used. The 'Tool Path Cornering' option allows for sharp or rounded corners to be created, in almost all instances a sharp corner is desirable. 'Move Control' is used if the g-code is being output from the Torchmate CAD in most cases this will be unused.

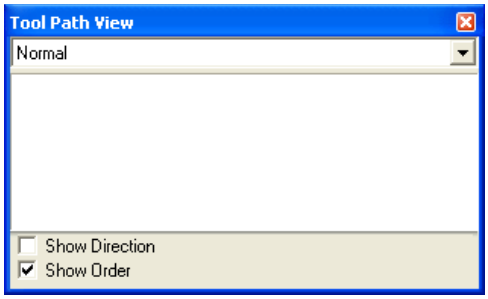
The third tab is 'Lead In/Out' and controls all aspects of lead ins and outs. The check boxes at the top enable a lead in, a lead out or both. For either type there is a choice between 'Arc' or 'Line'. When an 'Arc' is selected a radius can be specified this is the length of the lead away from the actual cut. For a line both a length and an angle need to be specified, this angle is in reference to the start point so a 90 degree angle will be perpendicular to the cut path. For lead ins an 'Online' lead can also be selected this is a lead in that will pierce on the part just a certain distance back from the start point; as the pierce is on the actual cut this type is generally not used. 'Overlap' is used to continue cutting past the start point before finishing a cut, this is used sometimes in oxy fuel cuts or with some plasma cutters that have a tendency to not finish a cut completely. Finally on this tab is 'Adjust Start Point' this allows for rough relocation of leads to the corner specified.

How do I reorder tool path cut order?

In Torchmate CAD there are multiple ways to reorder tool path cut order; each way is suited to certain circumstances. The cut order is generally edited after tool paths have been made, some of the methods however do allow for order to be specified prior to a tool path being created.

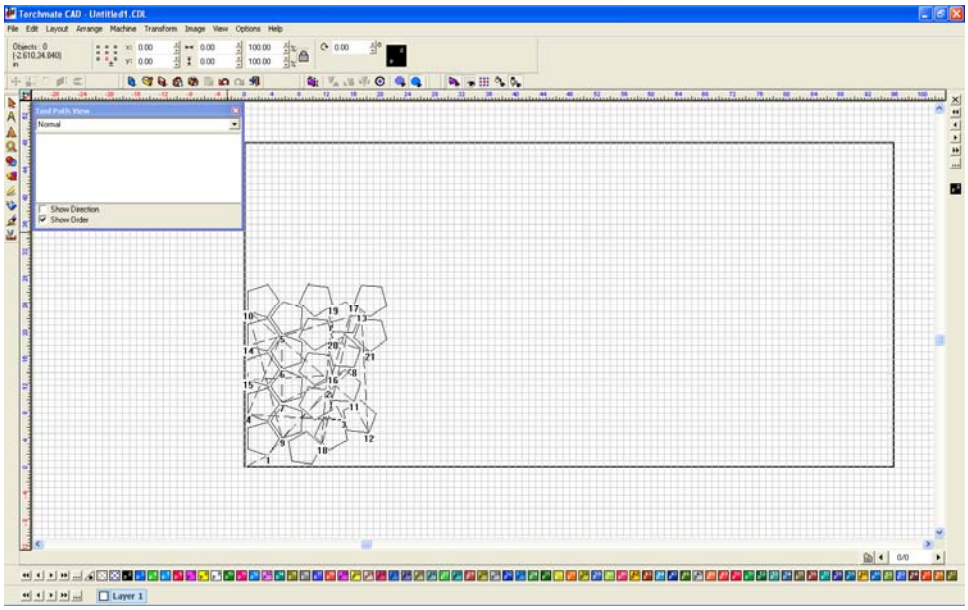
The simplest method of editing order is to use the 'Tool Path Viewer'. This method is simple to use but when doing a large amount of parts can become time consuming. To access this method go to 'View'→ 'Show Tool Path Viewer'.



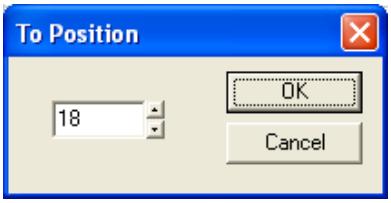


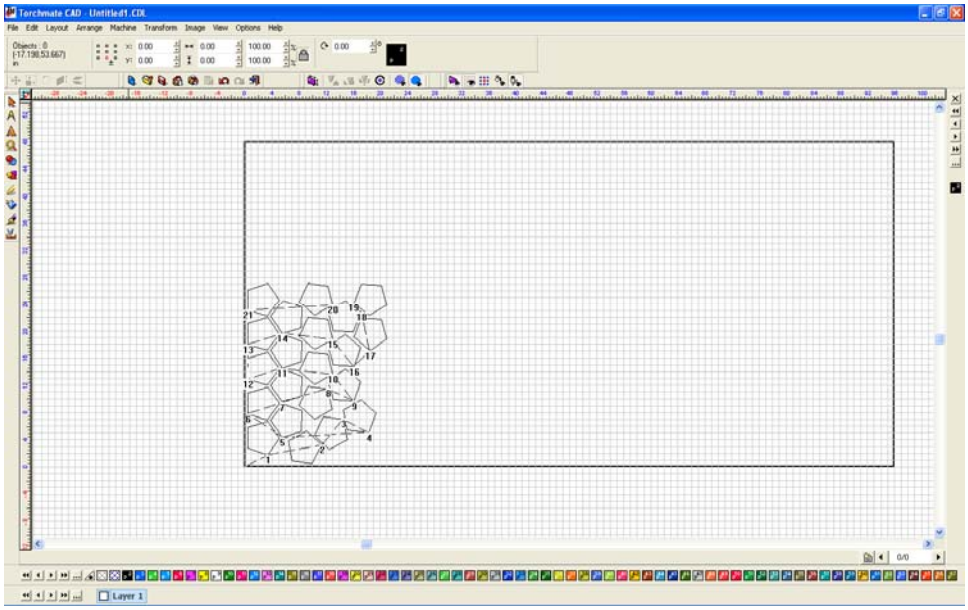
The 'Tool Path Viewer' opens a window, if 'Show Order' is checked it will display the cut order.

The cut order is displayed as numbers next to each part. When a part with multiple inside cuts is displayed it is still only displayed as a single number.

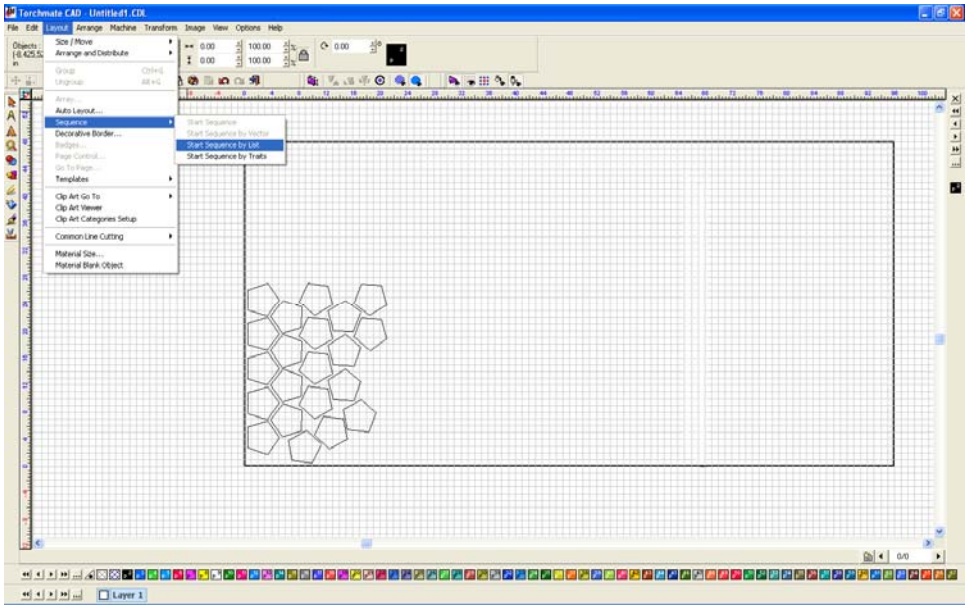


To change the order double click on any part, the ‘To Position’ window will appear. The number indicates the sequence position, simply type in the new position to change the order.

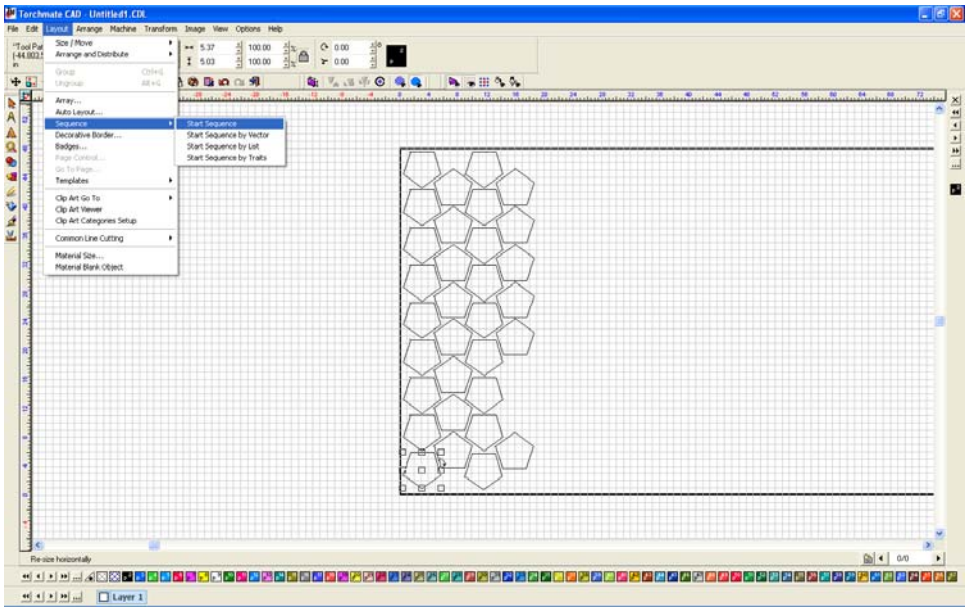




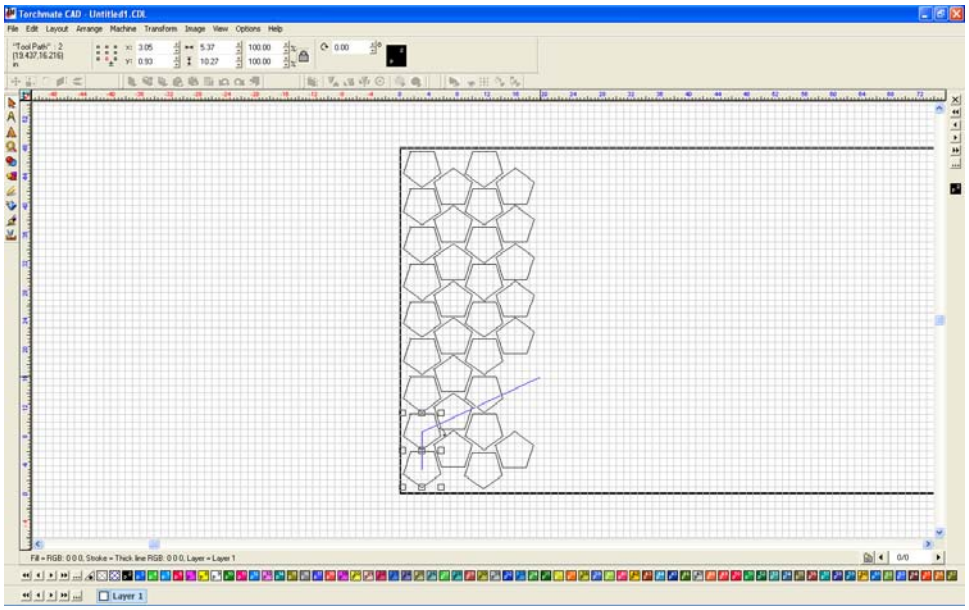
Another method to reorder cuts is called ‘Sequence’. There are four sequence options, three of which are quite useful in many cases; sequence by traits is only useful if each part is a different layer or color. To access the sequence options go to ‘Layout’ → ‘Sequence’ here the four choices are available.



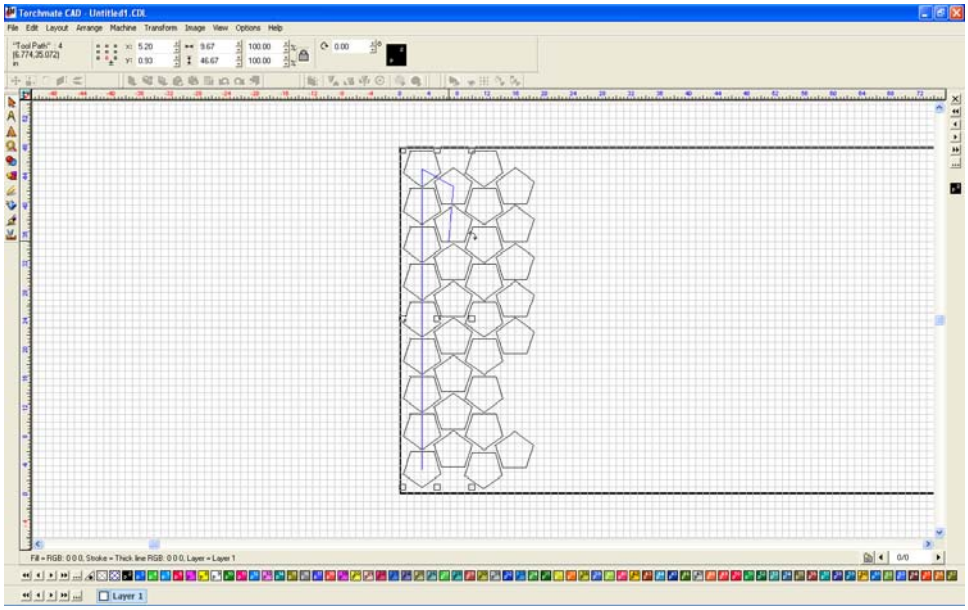
‘Start Sequence’ is the first choice in the menu. Begin by selecting the first object to be cut then select ‘Start Sequence’.



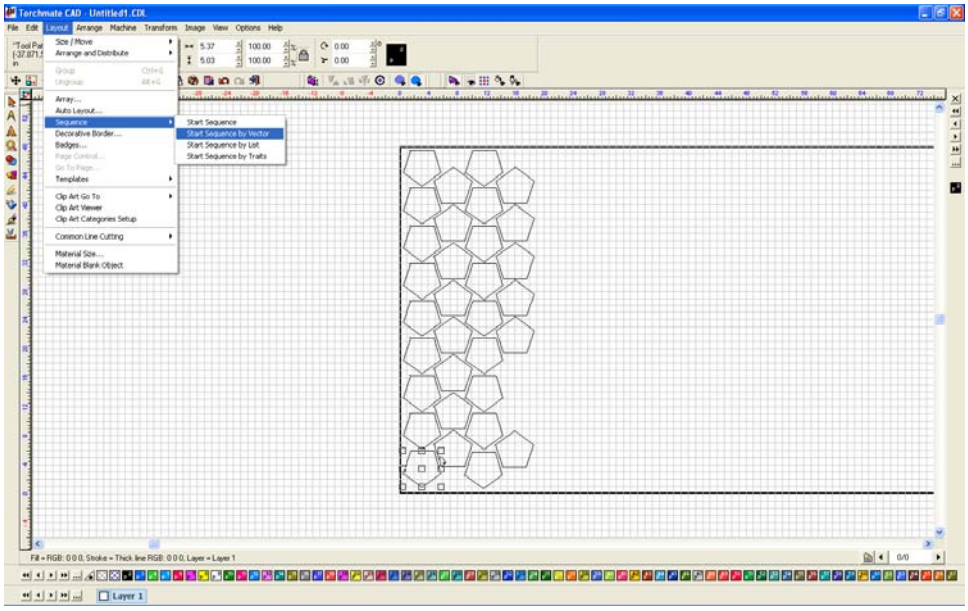
A blue line will emanate from the selected part. To make the cut out order click on each part in the order they need to be cut in.



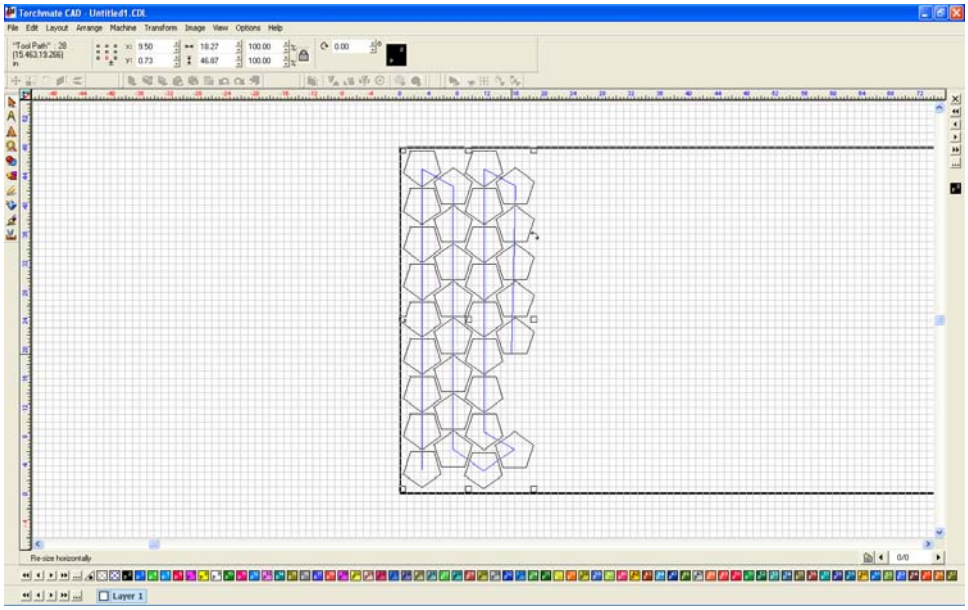
Continue doing so clicking on each part. For a large amount of parts this method can be a bit time consuming however for a few larger parts it is very quick and easy.



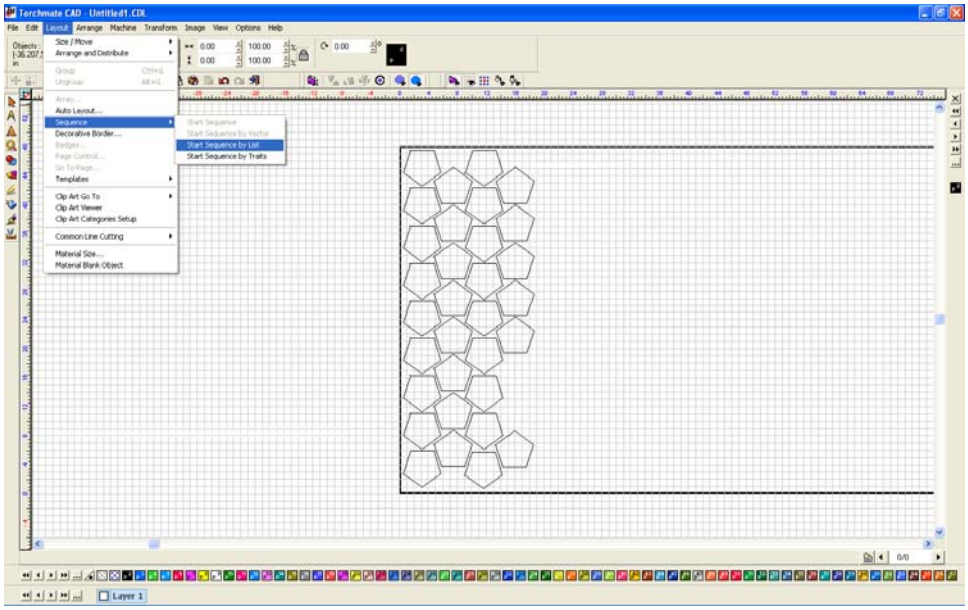
‘Start Sequence by Vector’ is similar to ‘Start Sequence’ but is more useful when rows of parts are used.



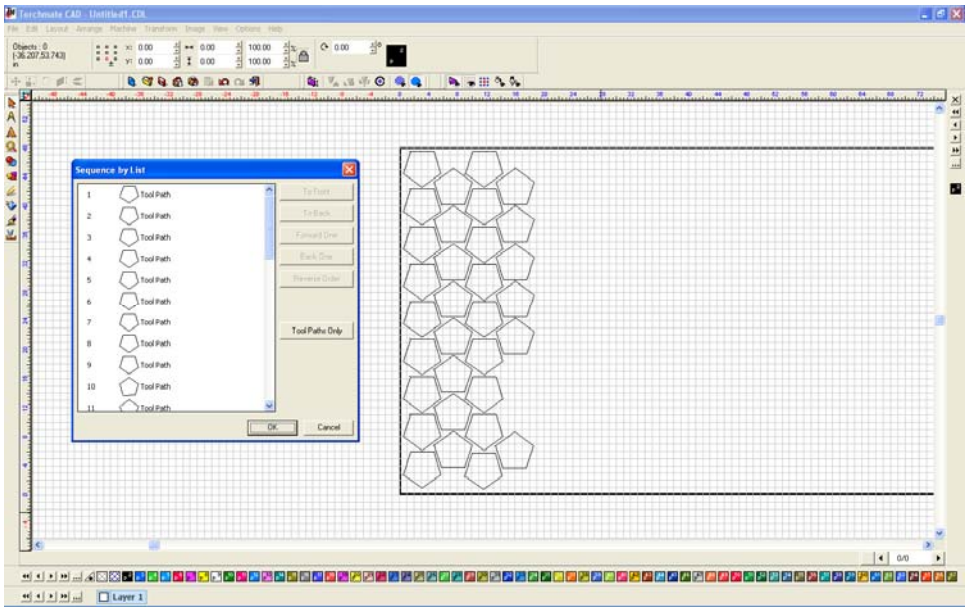
With ‘Start Sequence by Vector’ it is not necessary to click on each part to order them, simply by clicking on two parts will also order the parts in-between in series. This is useful for rowed parts since only the parts on the end of the rows need to be clicked to sequence the entire sheet.



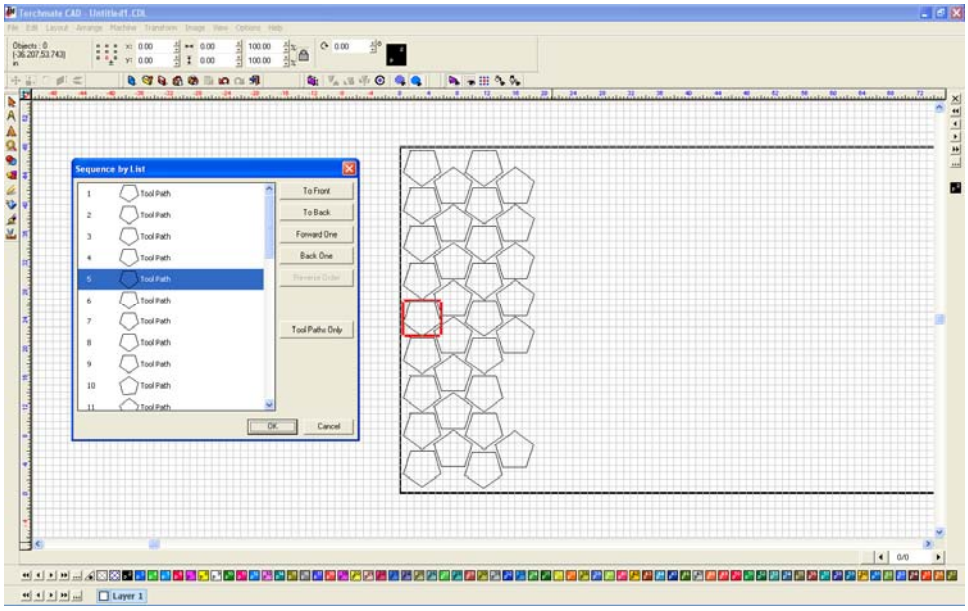
‘Start Sequence by List’ is another way to change cut order.



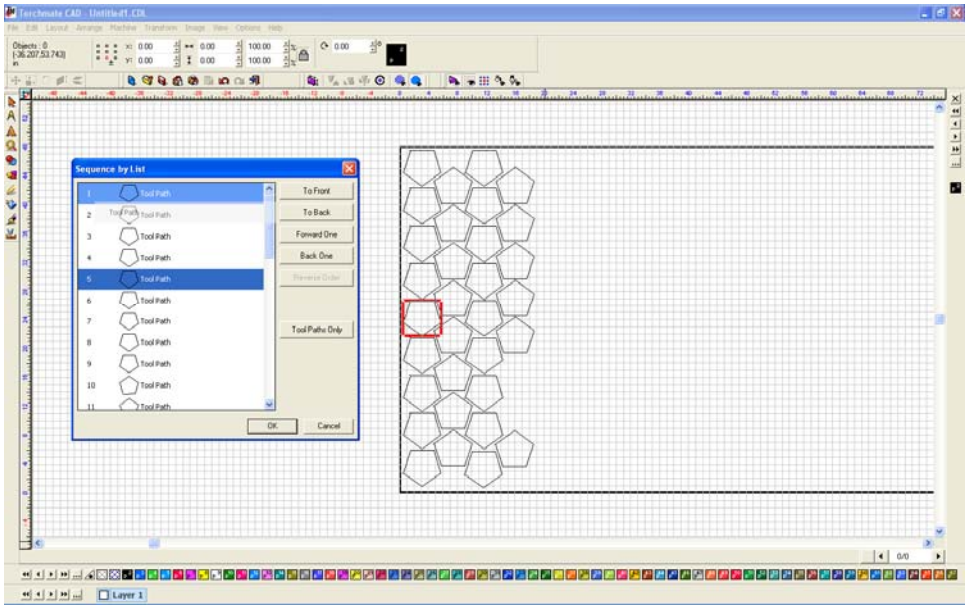
By list uses a moveable list to order cuts.



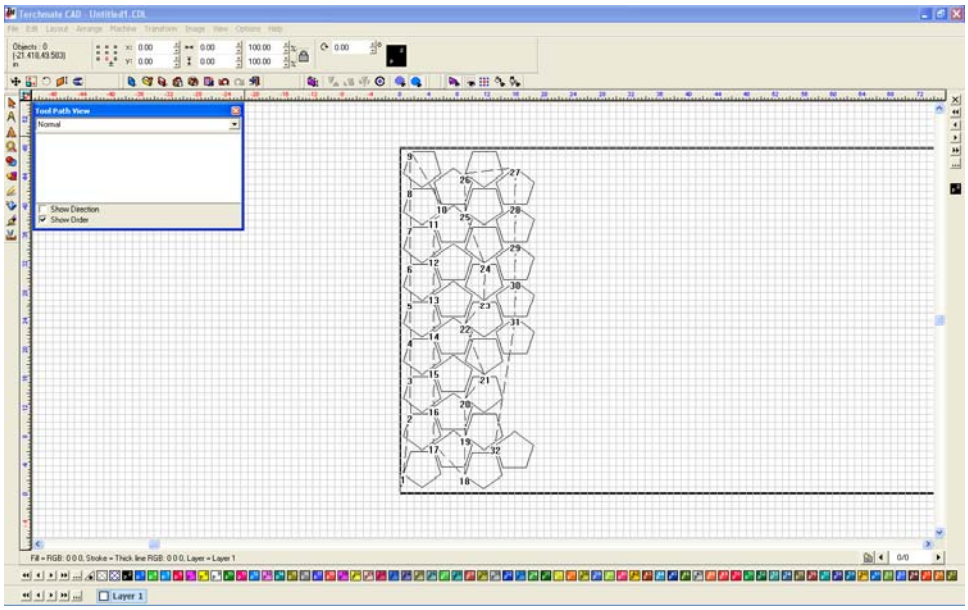
When a part is selected from the list it will also be highlighted with a red square on the work area.



The order is changed by dragging a part up or down the list, or by using the positioning buttons in the menu.



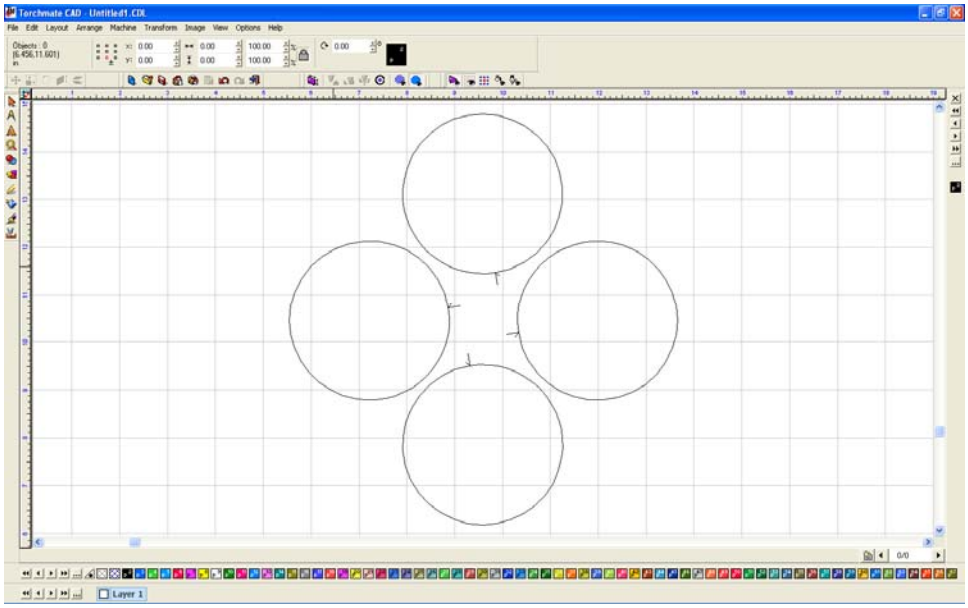
Any time the cut order is changed it can be viewed again by going to the ‘Tool Path Viewer’ in the ‘View’ menu. Keep in mind that the order displays a part with multiple cuts simply as a single number.



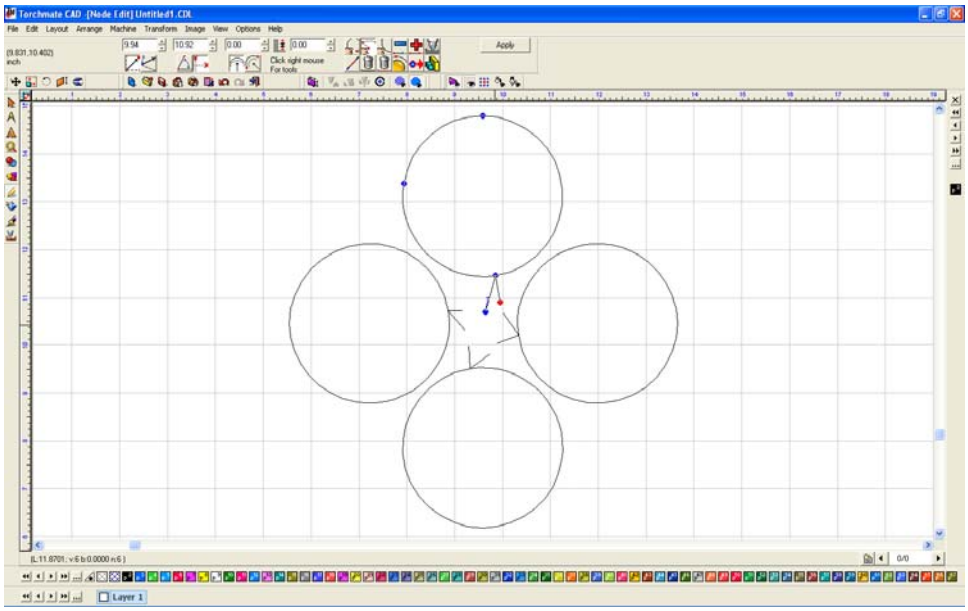
What is common line cutting and how is it accomplished in Torchmate CAD?

Common line cutting is a method of drawing that links some or all of the tool paths together as one single path. This is useful when working with thick plate using oxy fuel or when cutting with plasma and using an edge start method. In both these cases restarting a cut in the middle of the plate can take an upwards of 30 or more seconds or may be impossible. Common line cutting allows the tool to cut from one part to the next without turning off. Keep in mind that the path may cross over itself while doing this, with plasma this may cause the torch to go out and need to be restarted. Common line cutting uses a lead in and lead out for each part, with the lead out of one part connected to the lead in of the next. The most difficult part of common line cutting is positioning each part so that the travel distance between parts is minimized as cutting thick plate is generally at very slow speeds.

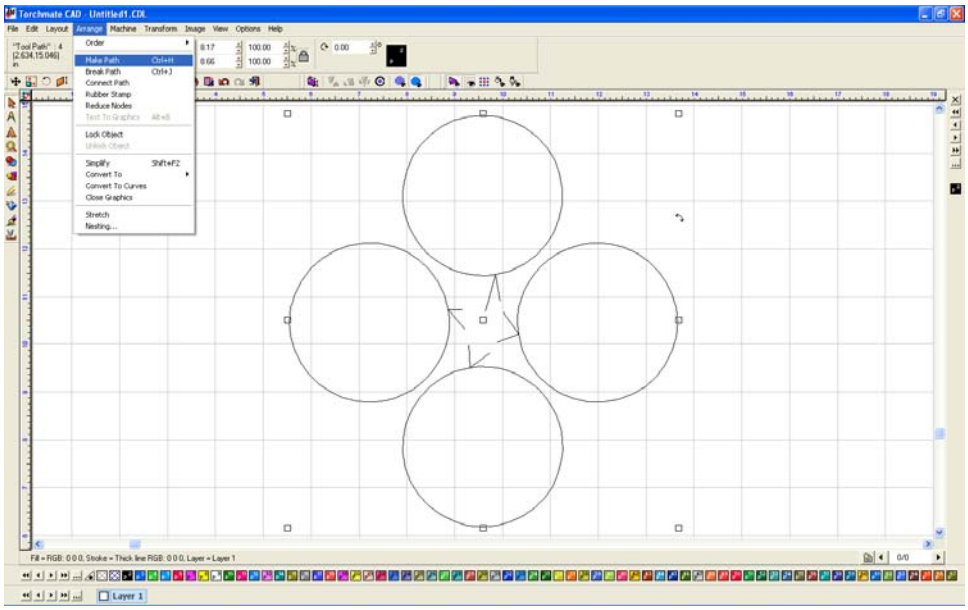
To common line cut begin by drawing the parts and laying them out on a sheet. Create a tool path with lead ins and lead outs, for the sake of clarity make them with different lengths or angles so they can be distinguished.



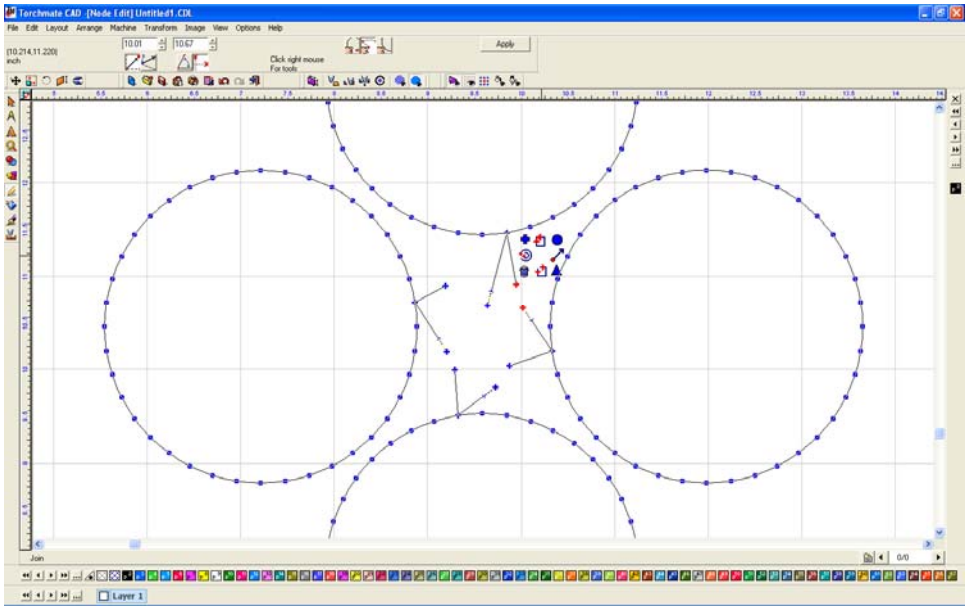
Edit each tool path so that the lead ins and outs are near eachother.



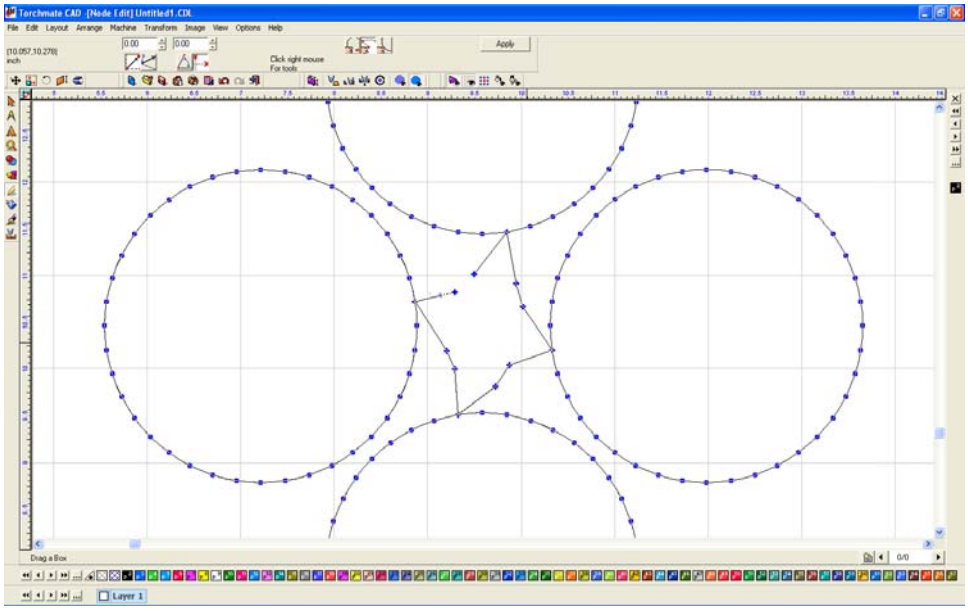
Select all of the tool paths, go to ‘Arrange’ → ‘Make Path’. This converts the tool path back to a polygon but allows the parts to be joined together.



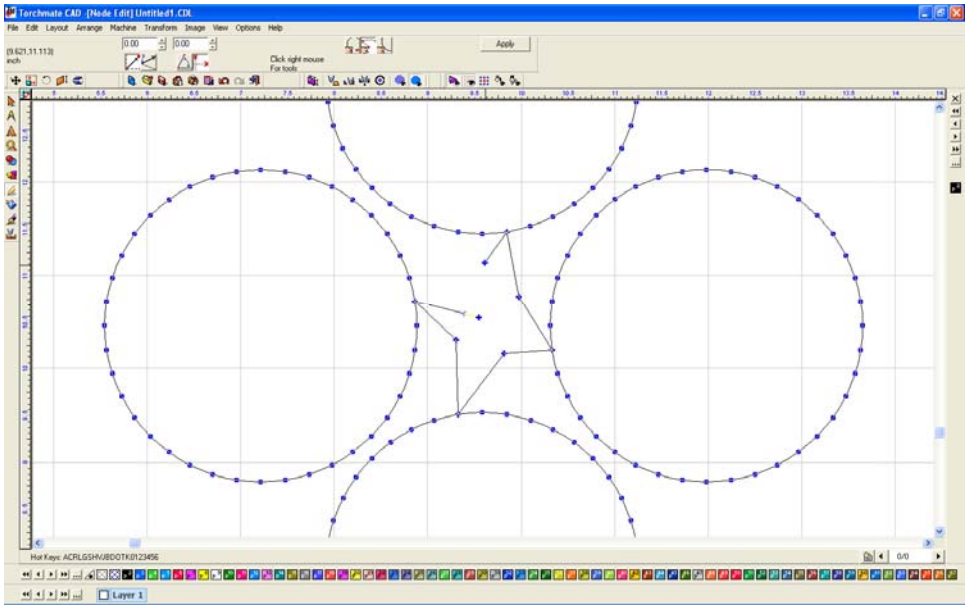
Double click on the parts to enter node editing. Join the lead in of one part to the lead out of another by selecting the two nodes, right-click, and select ‘Join’ (middle-top option).



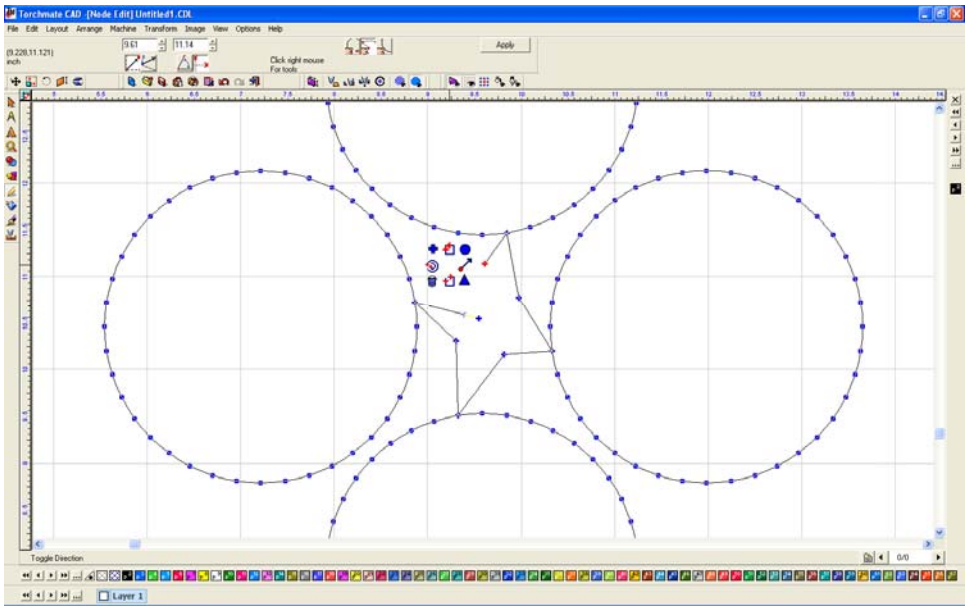
Repeat this with all of the parts leaving one lead in and one lead out not joined.



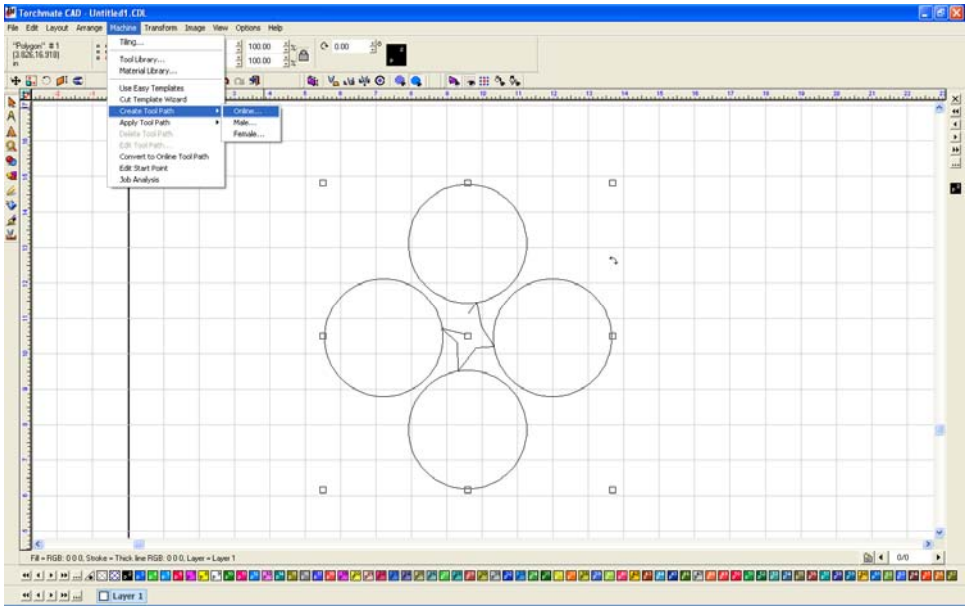
The nodes can be edited just as any drawing and the leads can be cleaned up by eliminating some of the nodes.



By joining the nodes occasionally the cut direction will change, to reverse it select the node that should be the starting point, right-click, and select ‘Set Start Point’ (the right-middle option).



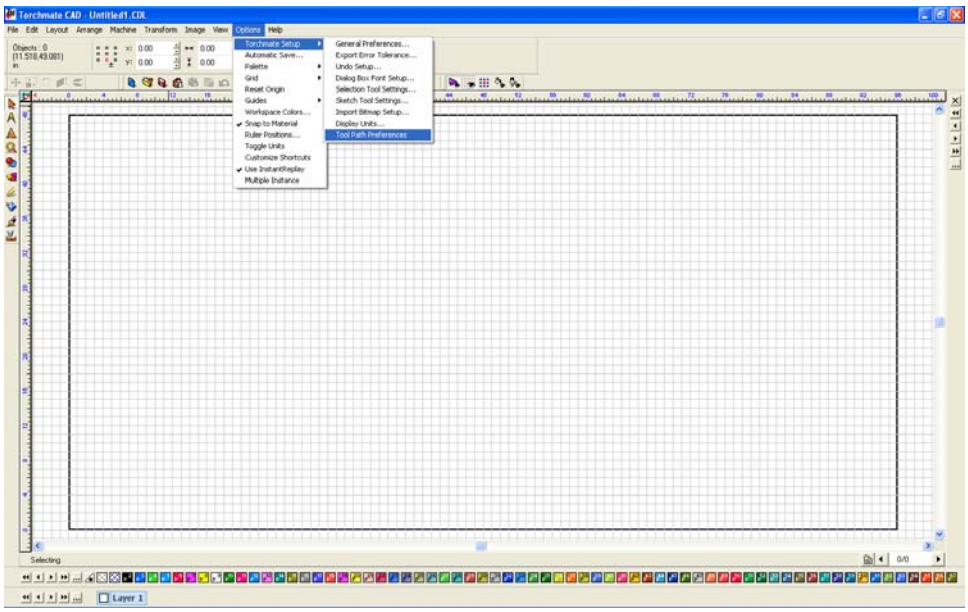
The parts are now joined; the last step is to create a tool path again. This time the tool path will be ‘Online’ since the offset was already done the first time the tool paths were created.



Again keep in mind when using common line cutting with plasma cutting the plasma can go out when traveling over an already cut area. When cutting parts with holes the holes will need to be created separately and cut order will have to be edited before cutting.

Why when I create a tool path the original drawing is grouped with the tool path and I cannot delete it?

In Torchmate CAD there is one default setting that must be changed to eliminate this problem. Go to 'Options' → 'Torchmate Setup' → 'Tool Path Preferences'.



Ensure that 'Group tool paths with originals' is unchecked. Press 'OK' and all future tool paths will be created just fine.