Version 6 CAD/CAM/CNC Tutorial
After today I will be able to:

- Draw a part in FlashCut CAD
- Use the Shape Library
- Make a toolpath in FlashCut CAM
- Generate a nest
- Generate G-Code
- Configure Fabrication Heads
- Run the CNC Controller to cut a part
- Use Advanced CAD/CAM Tools
Tutorial Lesson List

• Lesson 1 – Features, Installation and Licensing
• Lesson 2 – Drawing in CAD
• Lesson 3 – Making a Toolpath in CAM
• Lesson 4 – Using the Shape Library
• Lesson 5 – Importing an Image
• Lesson 6 – Fabrication Heads and Cut Charts
• Lesson 7 – Advanced CAD/CAM Tools
• Lesson 8 – More Advanced CAD Tools
• Lesson 9 – Cutting a Part in CNC
• Lesson 10 – Diagnostics and Help
Lesson 1 – Features, Installation and Licensing
FlashCut CAD/CAM/CNC
Version 6 Standard Features

• Simple, powerful 2-D CAD
• Integrated CAM with true shape nesting
• Automatic kerf compensation
• Automatic generation of lead ins and lead outs
• Automatic cut settings based on material
• Simple import of pictures and drawings
• Easy CNC program flow control including “Jump to Line”, forward and reverse processing
• Integrated Plasma Torch Height Control
• Industry renowned CNC control
Additional CAD Features

• V6.0 Standard:
  • Improved DXF/DWG Import
  • Measuring tool lets you determine any part dimension or distance on a drawing
• V6.0 Pro Package
  • Integrated Shape Library
  • DXF Export
  • Bring TT fonts directly into CAM without exploding
  • “Fix Tool” finds and corrects CAD mistakes instantly
  • Bridge internal cutouts instantly such as the center of an “O”
• Ability to curve text around any shape
• Ability to weld shapes together as one
Lesson 1 – Features, Installation and Licensing

Additional CAM Features

• V6.0 Standard:
  • Improved nesting performance
  • Editable plasma and oxy-fuel cut charts for Hypertherm and Thermodynamics
  • Support for Multiple Fabrication Heads including Plasma, Water Jet, Oxy Fuel, Spindle, Laser Pointer
  • Smart feedrate and THC handling of small holes
• V6.0 Pro Package:
  • Enhanced nesting interface of multiple parts and external files
  • Enhance production nesting features – fill sheet, multi sheet, partial sheet
  • Corner Looping
Additional CNC Features

- **V6.0 Standard:**
  - Marker Support
  - Improved THC
  - Advanced security to prohibit unwanted changes to configuration
  - Support for Simultaneous Multiple Fabrication Heads including Plasma, Water Jet, Oxy Fuel, Spindle
  - Feedhold on loss of Arc Transfer Signal
- **V6.0 Pro Package:**
  - Lead in automatically added when starting in the middle of a G-Code – minimizing blemishes on part
  - Ability to turn THC on and off in the middle of a cut.
  - Angular sheet alignment
Lesson 1 – Features, Installation and Licensing

Other Version 6.0 A-la-carte Features

• Laser Pointer support including finding zero and toolpath preview directly on sheet
• Enhanced communications with Hypertherm Powermax plasma for setting amperage, pressure and reporting status
• Bar Code input and automatic file loading for streamlined production
• Advanced Input Configuration including Joystick Jogging
• Dynamic control of ventilation zones
<table>
<thead>
<tr>
<th>Feature</th>
<th>Standard Package</th>
<th>Pro Package</th>
<th>Available A-la-carte</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simple, powerful 2-D CAD with Improved DXF/DWG Import</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Measuring tool lets you determine any part dimension or distance on a drawing</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Integrated CAM with true shape nesting includes Improved nesting performance</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Editable plasma and oxy-fuel cut charts for Hypertherm and Thermodynamics</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Support for Multiple Fabrication Heads including Plasma, Water Jet, Oxy Fuel, Spindle</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Smart feedrate and THC handling of small holes</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Automatic kerf compensation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automatic generation of lead ins and lead outs</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Automatic cut settings based on material</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Simple import of pictures and drawings</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Easy CNC program flow control including &quot;Jump to Line&quot;, forward and reverse processing</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Integrated Shape Library</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DXF Export</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Bring True-Type fonts directly into CAM without exploding</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Fix Tool finds and corrects CAD mistakes instantly</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bridge internal cutouts instantly such as the center of an &quot;O&quot;</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Enhanced nesting interface of multiple parts and external files</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Enhance production nesting features - fill sheet, multi sheet, partial sheet</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Corner Looping</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Ability to Turn THC on and off while cutting</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Curved text along any shape</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Shape welding and general boolean operations among shapes</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced file management including bar code scanning support and automatic file loading</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Laser Pointer and Laser Pointer Software Support</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Laser Pointer Software Support Only</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dynamic Ventilation Zones</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Installation

1. Launch the FlashCut CNC Version 6 Install Icon
2. Choose “Run” in the following:
• If you have a previous version, it will be automatically uninstalled
Lesson 1 – Features, Installation and Licensing

- The new software will then be installed
Choose the folder location for the software to install
• The installation is now complete
Licensing

1. Each system comes with two software licenses:
   - One license embedded in the controller for any PC connected to the controller.
   - One “software only” license which can be activated on any PC regardless of being connected to the controller
2. Licensing will allow you to save CAD/CAM files, Create G-Code and send G-Code to the controller.
3. An unlicensed program will run in “Evaluation Mode” and will allow the user to play with the CAD/CAM features and create G-Code of 25 lines or less
4. To activate a “software only” license, first go to the “License” Button
5. Make sure you have an internet connection with the firewall off or with settings that will allow for an outside program to write to your hard disk.

6. Choose the Activate PC License button.

7. Copy the License Key form the licensing e-mail and paste it in the License Key text box by typing or using the Paste from Clipboard option.

8. Choose the License Software button.
The features that you are licensed for are noted under a green key.
Each purchased license is good for one PC and one controller.
The PC license can be transferred to a new PC by first selecting the Deactivate PC License button on the original PC and then re-activating the license on a new PC using the steps above.
You must have an internet connection for this to work.
1. Once you have activated your license, can now view the list activated features on the license panel.

2. If you have a green key, then you have activated the features. If there is instead a grey lock, then that portion is in evaluation mode.

---

**Lesson 1 – Features, Installation and Licensing**
1. To activate the controller license you must be connected to the controller:
   1. Make sure your USB cable is connected and your controller is turned on.
   2. Go to the CNC tab
   3. Hit the Connect button – which will turn green when connected.

2. Once you are connected to the controller, it is not necessary to do a “software only” license unless you want to license a second PC or if the controller PC is not always going to be connected to the controller.
Lesson 2 – Drawing in CAD
Lesson 2 – Drawing in CAD

Create Tools

- Lines
- Rectangles
- Arcs
- Circles
- Text
- Points
- Polygons
- Elliptical Arcs
- Ellipses
- Splines
Lesson 2 – Drawing in CAD

Shape Library

Shapes
Lesson 2 – Drawing in CAD

Modify Tools

- Chamfer
- Fillet
- Extend
- Trim
- Scale
- Offset

- Fix Drawing
- Cut
- Explode
- Group
- Ungroup
- Ungroup All
Lesson 2 – Drawing in CAD

Transform Tools

- Copy
- Delete
- Move
- Rotate
- Measure
- Bridge
- Mirror
- Linear Matrix
- Circular Matrix
Lesson 2 – Drawing in CAD

Toggle Snap Tools

- Snap to:
  - Center
  - Endpoint
  - Grid
  - Horizontal/Vertical
  - Intersection

- Mid-point
- Nearest
- Parallel
- Perpendicular
- Quadrant
- Tangent
Lesson 2 – Drawing in CAD

Importing Files

- Image Outline / Silhouette of: bmp, jpg, png, ras, tiff
- DXF/DWG Drawings
- Image Centerline
Lesson 2 – Drawing in CAD

Making a Bracket
Lesson 2 – Drawing in CAD

Make a 6” x 6” square

- Choose the Rectangle tool
  - Either Select opposite corners with the mouse
  - OR Set X Y Coordinates of both corners in the dialog box
  - “Check it in” with the green check
Lesson 2 – Drawing in CAD

Make a polyline

- Choose the Line tool
- Choose the “Continuous Line” option
- Select 3 points (4, 6), (4, 2), (0.2) with the mouse or type XY values followed by the Green Check
- Hit the Red “X” or the Esc key to end the polyline
Draw a circle

- Select the Circle Tool
- Select the center point (1, 1) with your mouse or type the XY center position in the dialog box
- Set the diameter with your mouse or the dialog box on the right
Lesson 2 – Drawing in CAD

Copy the circle

- Select the existing circle – It will turn red
- Select the copy icon
- Select the center of the existing circle as a reference point
- Select the new center point (5, 5) or (4,4) offset from ref.
- Check it in
Lesson 2 – Drawing in CAD

Trim the excess

• Select the Trim tool
• Hover your mouse over the parts you want to trim until it turns red, then click to remove
Lesson 2 – Drawing in CAD

Create a Fillet

- Select the corner or two lines to fillet – They will turn red
- Choose the Fillet Tool
- Adjust the radius with your mouse or enter a radius of 1.0
- Check it in
Create a Chamfer

- Select the corner or two lines to chamfer – They will turn red
- Choose the Chamfer tool
- Either drag to size or enter a length of 1.0 on the Distance 1 & 2
- Check it in
Lesson 2 – Drawing in CAD

Save the Drawing

- Select the Save icon or from the File menu choose Save
- Type the name of the drawing “Bracket” in the dialog box
- Hit the Save button
Lesson 3 – Toolpath in CAM

- Hit the double arrow button between CAD and CAM
- An initial toolpath will be generated with your default parameters.
Lesson 3 – Toolpath in CAM

Selection Tool

- Used to select groups of entities
  - All Breaks
  - Cutout Breaks
  - Perimeter Breaks
- Ctrl-mouse select will also select and deselect multiple entities

- All Parts
- All Operations
Display Options

- Convenient way of displaying and hiding:
  - Cutting direction
  - Kerf compensation
  - Perimeter of material
  - Parts
  - Rapid Moves
  - Toolpath
Lesson 3 – Toolpath in CAM

G-Code Generation

- Used to Generate and Edit G-Code
Lesson 3 – Toolpath in CAM

CAM Actions

• Project Settings
• Fabhead (Plasma) Settings
• Grid Nesting

• True-Shape Nesting
• Cutting Order
• Reset Tool Path
Lesson 3 – Toolpath in CAM

View Tools

- Sheet Number
- Pan
- Zoom
- Zoom Window
- Zoom Workpiece Extents
- Zoom Toolpath Extents
Lesson 3 – Toolpath in CAM

Setting Lead-Ins and Lead Outs

- Select Cutout Breaks
- Edit Lead-In parameters - Type: Arc; Radius 0.125; Angle: 120
- No Lead-Out, No Overburn (=0.0)
- Move lead in to desired location by dragging breakpoint
- Check it in

[Diagram showing Lead-In and Breakpoint]
Lesson 3 – Toolpath in CAM

Setting Workpiece Dimensions

- Zoom to Workpiece Extents
- Select outline of Sheet or Project Settings icon
- Set to 48 wide x 24 long
- Make sure Default Cutting Head is correct and set material properties
- Check it in Sheet outline
Lesson 3 – Toolpath in CAM

True-Shape Nesting

- Make 30 brackets, 0.25” apart and 1.0” away from edge of material
- Hit the Start Nesting button
- Check it in
Lesson 3 – Toolpath in CAM

Save the CAD-CAM File
Lesson 3 – Toolpath in CAM

Send G-Code to Machine

- Hit the double arrow button between CAM and CNC
- G-Code will be generated for the toolpath in CAM and the CNC panel will automatically open. (Note: If you hit the CNC button, the CNC panel will open, but it will not load the G-Code for the current toolpath in CAM.)
Lesson 4 – Shape Library
• Select the New File icon or select “New” from the File menu
Select the Shape Tool

- Select the Shape Tool
Lesson 4 – Shape Library

Customize a Shape

- Scroll down and choose the shape that you want
- Edit the shape parameters in the right dialog box
- Choose a Pivot Point location and move pivot point with mouse to desired location or type insertion point X Y values
- Hit the green check
Save the Drawing

• Select the Save icon or from the File menu choose Save
• Type the name of the drawing “Pulley Guard” in the dialog box
• Hit the Save button
Lesson 5 – Toolpath from Image
Choose Image to Convert to Vectors

- Choose the Import Silhouette Image Icon (Make sure it is activated in the Configuration…CAD…Features)
- Browse to an image file (bmp, jpg, png, ras, tiff). Note the larger the image is, the more accurate the conversion will be.
- Determine Scale % or absolute XY dimensions, XY Position and other import options
- Check it in Lesson 5 – Importing an Image
Lesson 5 – Importing an Image

Edit the geometry

- Ungroup the imported geometry to make it editable
- Delete, add or modify any geometry that needs to change - In this case, eyes to ovals.
- Note that using the Alt-Mouse select on any segment will “Chain Select” the entire perimeter.
Lesson 5 – Importing an Image

Create the Toolpath in CAM

- Hit the CAM button to automatically create a toolpath
- Hit the “Cut the Parts” icon to generate G-Code and send it to the CNC panel
Lesson 5 – Importing an Image

Cut the Parts in CNC
Lesson 6 – Fabrication Heads and Cut Charts

• Fabrication heads (Fabheads) are the actual cutting heads on your machine.
• You can have more than one installed on your machine.
• Plasma, Water Jet, Oxy Fuel, Router, Marker, Laser or Laser Pointer.
• Some have associated Cut Charts with cutting parameters such as feedrate, kerf width and cut height which vary for a given type and thickness of material.

![Fabrication Heads and Cut Charts Interface](image-url)
Lesson 6 – Fabrication Heads and Cut Charts

**Adding a Fabhead**

- Go to the Configuration Screen by hitting the “gears” button
- Select “Fabrication Heads” and “New Fab Head.”
Adding a Plasma Fabhead

- Choose the “Type”, SubType” and “Model” from the drop-down menus. “ID” and “Name” will be automatically populated, but you can edit them.
- Type an XY offset if you have more than one Fabhead.
- Select RS-232/485 and a Com Port if you have direct communication with the plasma for setting Current and Pressure on the fly.
- Fill in the proper M codes for turning the Torch on and off. Typically these are 50 and 51 respectively. Note that the M codes must be defined in the M-Code Definitions.
- Fill in the Automatic M-Codes or Macros for turning the torch and THC on for cuts and off for rapids. Note that the Macros and M-Codes must be defined in the M-Code Definitions and Execution Screens.
- Hit the “Add Fab Head” button to save your Fabhead changes when you are done.
- Save the Setup file.
Lesson 6 – Fabrication Heads and Cut Charts

Adding a Plasma Fabhead (THC)

• If you have Torch Height Control, check the Box and fill in the parameters:
  • Arc Voltage Divisor – This is typically 50 and must match the exact voltage divider setting in your torch. Note that in order for the torch height control system to work properly, \[(\text{maximum raw voltage}) / (\text{arc voltage divisor})\] must be smaller than 6V.

• Lock Out Factor - This will turn the THC off to prevent unwanted diving when the XY velocity of the torch is below this percentage of the programmed federate.

• Lock Out Hysteresis - The THC will turn back on again when the XY velocity goes above the \[((\text{Lock Out Factor}) + (\text{Lock Out Hysteresis})) \times (\text{Programmed Feedrate})\].

• Set Point Offset – Offsets the small residual voltage when torch is not in use.

• Sensitivity - This is a factor that determines how sensitive the THC acts towards voltage fluctuation. This number can vary from 1-64 where a 1 is the most sensitive and a 64 is the least sensitive.

• Use Kerf Crossing Detection – Will temporarily turn off THC while crossing a previous cut on the sheet to prevent unwanted diving.
Adding a Plasma Fabhead (Sensing)

- Sensing will find the surface of the plate either before each cut or to find program 0.
- Choose the sensing direction and feedrate.
- Choose the Input Line that will be used for the primary sensor. This is typically an ohmic sensor.
- Sensor Offset is the extra distance that the Z axis travels in the upwards direction after the sensing the sheet surface. This distance can account for any hysteresis or slop in the sensing switch or Z Axis mechanical system.

- Use Additional Sensor – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.
- Program Zero Sensing – Check the box for the ability to seek the sheet surface to set program zero.
- Max distance to Move – The maximum travel distance to find the sheet.
- Retract Distance – The amount that the Fabhead will move up after sensing the sheet.
- Touch Off at Start (G605) – Senses the sheet at the start of each cut.
- Sensing Limit – The lowest coordinate that the Z axis will sense to.
Lesson 6 – Fabrication Heads and Cut Charts

Plasma Fabhead Advanced Control

- Cut lead-in when returning to toolpath – Designed for recovering from a lost torch using the following steps:
- Use jump to line and use the run forward, backward and feedhold controls in “dry run” mode to the point the cut stopped.
- Jog off of the toolpath to a point that will be off the finished part.
- Take it out of “dry run” mode and start the program from that point.
- The system will use that point as a temporary pierce position and then cut a lead-in to the last feedhold position on the part.
- The program will proceed normally from that point.
Some of the Plasma and Oxy Fuel Fabheads have pre-loaded cut charts with recommended settings such as feed rate, pierce height, cut height and THC set point.

You can use the Standard Cut Charts for the Fabhead manufacturers’ recommended settings.

If you choose the Custom Cut Chart option, then a *.csv file copy of the standard cut charts will be stored into the specified location when you hit the “Copy Standard Charts To Folder” button. Any value in these cut charts can be edited using a spreadsheet program like Excel.
Lesson 6 – Fabrication Heads and Cut Charts

Adding a Laser Pointer

- Go to the Configuration Screen and Select “Fabrication Heads” and “New Fab Head.”
Adding a Laser Pointer (cont.)

- Choose the Type as “Laser Pointer”
- “ID” and “Name” will be automatically populated, but you can edit them.
- Type the X and Y distance of the center of the laser pointer from the center of the Plasma Cutter in the X and Y offset fields.
- Fill in the proper M codes for turning the Laser Pointer on and off. Typically these are 10 and 11 respectively. Note that all M codes must be defined in the M-Code Definitions.
- Since a laser pointer is a simple device, there is probably no need for a macro, so use the same M codes you used in the Laser Control in the Automatic Execution Mode.
- Hit the “Add Fab Head” button to save your changes.
Lesson 7 – Advanced CAD/CAM Tools

- Text Tool
- Fix Drawing
- Measure Entities with Ruler
- Bridge Entities
- Corner Looping
- Nesting Parts from Multiple Files
Open the Text Tool

• Choose starting point with your mouse or type the XY Positions
• Type the text you want
• Choose a font. Note that once you choose an initial font, you can use the up and down arrows to view other fonts.
• Tailor the size, kerning, line spacing, angle, bold, and italic options
• Check it in
Create the Toolpath

- Go to the CAM mode by selecting the double arrows.
- Note that the toolpath is created without the need to explode the text. (Unless you need to modify the geometry of the text such as adding bridges)
- You can move and change the lead in of each letter which will be retained even if another letter in the word is edited.
Fix Tool

• Go back to CAD and using the line tool, draw a box around the text, but don’t connect the last point with the first point – This is easier to do if you deactivate all of the snapping tools.
Try to Create the Toolpath

- Go to the CAM Tab by hitting the double arrows
- The toolpath for the rectangle did not appear because it is not a closed entity.
- Note that if you choose the “Make the Top Level a Cutout” Feature Type button, the rectangle will appear, but the text will become a cutout and not a part.
- Go back to “Make the Top Level a Part” Feature type
Lesson 7 – Advanced CAD/CAM Tools

“Fix” the Rectangle

- Go back to the CAD tab
- Choose the “Fix Drawing” icon
- Chose a tolerance larger than any of your drawing errors.
- Hit the “Search Issues” button
- Any issues found will be noted with a green circle. Any issues that cannot be fixed automatically will be noted with a red circle
- You can either choose the “Fix All” button to automatically fix the issue or fix the issues yourself.
Choose the “Measure Entities with Ruler” Tool
Select the mode you want: Points, Entities or Protractor
Select the portion of the drawing you want to measure
The corresponding dimensions will appear in the right side.
Create the toolpath

- Go back to CAM by clicking on the double arrows
- Note that the rectangle now appears no matter which feature type is the top level
Using Bridges

- Go back to the CAD tab
- Draw the text “BOA” surrounded by a rectangle
- Note that if you cut this part, the centers of the B, O and A (shaded in blue for illustration) would fall out.
Bridge the Interior Features

- Select the text and explode it using the Explode button so that it becomes line segments.
- Choose the Bridge Tool
- Span Bridges on the B, O and I to join the interior of the letters to the surrounding box.
- The bridge will form when you hit the green check or when you start a new bridge.
- Note that you can set the width of each bridge individually and you can move the bridge with your mouse before it is finalized.
Create Corner Loops

- Corner looping minimizes dross on sharp, exterior corners.
- Hit the double arrows to go to CAM mode
- Go to the Plasma Settings icon
- Set Corners to “Rounded Loops” or “Triangular Loops”
- Set the Corner extension distance, for example 0.15 in.
- Hit the Green Check
- Save the CADCAM File
Nesting Multiple Files

- You can not only nest the shapes within a given CADCAM file, but you can reference external CADCAM and DXF files in your nest.
- Select the Project Settings icon
- Specify the size and material properties of the sheet you will be cutting. In this example we will use a 48” x 24” Mild Steel Plate.
Nesting Multiple Files (cont.)

- Choose the Import External Files icon
- Choose the files that you want to import and then hit the Green check for each one.
- In this case we are nesting all files made in the different lessons.
Lesson 7 – Advanced CAD/CAM Tools

Nesting Multiple Files (cont.)

- Select the True Shape Nesting icon
- On the right will be a list of the parts to nest. Click on each name and change each count in the text box below. Note that the named part gets highlighted in blue.
- Check “Automatically add workpieces” if you need more than 1 sheet.
- Fill in the other options such as nesting direction, min. distances, etc.
- When you are ready hit the Start Nesting button. Note that larger nests with more complex geometry will take longer to nest.
Lesson 7 – Advanced CAD/CAM Tools

Send Nested Sheet to CNC

- Select the sheet that you want to cut
- Select the double arrow between CAM and CNC
- The G-code of the nested toolpath will be sent automatically to CNC
Lesson 8 - More Advanced CAD Tools

- Elliptical Arc
- Mirroring
- Curved Text
- Welding
- Linear Pattern
Elliptical Arc

- In CAD Choose the Elliptical Arc tool
- Choose the center point for the full ellipse with your mouse or by typing on the right parameters panel.
- Choose the major and minor radii of the full ellipse
- Choose the start angle and end angle with your mouse or by typing on the right.
- Hit the Green Check Box.
Mirroring

- Select the Mirror tool
- Check the box to copy the objects. Otherwise, it will just flip the object.
- Select the elliptical arc to mirror and right click to accept it.
- Select the left end of the elliptical arc as one endpoint of the mirror line.
- Select the right end of the elliptical arc as other endpoint of the mirror line.
- While you are selecting you will see the mirror line appear as a dotted line.
- The mirrored object will now appear.
Curved Text

- Using the text tool. Type the text that you want to curve in any font and size.
- Select the Curved Text Tool
- Select the text that you want to curve then right click to confirm.
- Select the curve that you want the text to curve around and right click to confirm. Note that if you hold down the “alt” key while you select, it will select the entire chain of geometry.
Curved Text (cont.)

- Adjust the curved text parameters to flip the text, determine the alignment and offset.
- In this case, we want the bottom of the text to be just below the top of the curve.
- Use the arrows next to the text to move the text along the curve.
- Note that you can still edit the text in the text editor.
Curved Text (cont.)

- Adjust the curved text parameters to flip the text, determine the alignment and offset.
- In this case, we want the bottom of the text to be just below the top of the curve.
- Use the arrows next to the text to move the text along the curve.
- Note that you can still edit the text in the text editor. In this case, we made the text bold.
Shape Welding

- Explode the text into geometry before welding by selecting the text and hitting the explode button.
- Select all of the geometry that you want to weld together. In this case it is all of the geometry so you can use “ctrl” A to select all or swipe over everything with your mouse.
- Choose the Weld Selected Entities tool to weld everything into one continuous shape.
Linear Pattern

- Draw a long horizontal rectangle and a short vertical rectangle on the left (see the red rectangle below)
- Select the red rectangle by using the "alt" key and the left mouse click.
- Choose the Linear Pattern tool and vary the parameters for the spacing and count.
- Note that there is also a Circular Pattern tool.

Lesson 8 – More Advanced CAD Tools
Welding and other Boolean Operations

- When you are happy with the lace pattern, you can weld them together by selecting them all and using the Weld Entities tool.
- Note that the Boolean Operations tool allows you to do welding and other operations such as subtracting entities and welding cutouts.
Lesson 9 – Cutting a Part in CNC

- Make sure the controller is connected by selecting the Connect Button (Note that if you are in demo mode or you are not connected to a controller this will not be an option)
- Agree to the “Safety First” message and hit OK if it asks you to upgrade the firmware.
- If you have limit/homing switches configured, home the machine by selecting the Home button and then “All”. The machine will seek the switches on all axes and set all of the Machine Coordinates to 0.
Set your Program Zero

- Jog the machine to the zero point of your program by clicking on any of the jog buttons either in “continuous” mode or any of the discreet distances. Note that the slider bar at the bottom determines the jog speed.
- Set the X and Y Program Coordinates to 0 by clicking on the “Set” button and “Zero X and Y”
- If you have a sensor on your Z axis such as ohmic or a switch, then set your Z axis coordinate by clicking on the “Set” button and “Sense Z Zero”. If you do not have a sensor, you can select “New...” under “Set” to set your coordinates.
Lesson 9 – Cutting a Part in CNC

Cut the Part

• Before cutting a part, you can verify the toolpath by first checking the “Dry run” option and then selecting the Green “Run G-Code” button.
• If everything looks good, you can move back to program zero by switching from Jog mode to Point Mode and selecting the Green Program Zero button.
• To cut the part, remove the “Dry Run” check box and select the green “Run G-Code” button.
• You can stop the program at any time by choosing the red “Feed Hold” button.
Lesson 9 – Cutting a Part in CNC

**Program Flow Control**

- The green “Run G-Code” button runs the program forward and cuts the parts.
- The red “Feed Hold button ramps down the motors and stops the program.
- The gray “Run G-Code in Reverse” button will trace the toolpath backwards, but will not cut.
- You can speed up or slow down the feedrate on the fly by hitting the + or – feedrate buttons.
- You can see the progress of the program by following the Yellow Dot on the Viewport or the Green Highlight of the current line of G-Code.
Lesson 9 – Cutting a Part in CNC

Jump to Line

• You can jump to any line of the program while the machine is idle by pressing the “Jump To Line” button. The Program Listing Box will turn blue.
• You can now select any G-Code line in the Program Listing Box or any line or arc segment in the Viewport.
• Select the “Execute Jump To Line” button
• A dialog will pop up verifying the Jump to Line was Completed. Choose “Yes” to move the machine to the start of the line you want to jump to.
• You can now run the program forward or backwards from that point by using normal program flow control.
Lesson 10 – Diagnostics & Help

• The Diagnostics Panel is accessible through the Magnifying glass icon.
• Live status of “Input Lines” is shown by the switch icons and blue LED’s (only active if the software is “Connected” to the controller)
• Live status of the Output lines are also available. You can turn any output line on and off simply by clicking on the corresponding button.
Help Screen and User’s Guide

- The Help screen can be accessed by clicking on the “?” button.
- This shows the current Software and Firmware versions.
- “Build Support File” button will create a zip file with all of the configuration settings and a log of past commands. This can be e-mailed to our support staff for in-depth diagnostics.
- Any online manuals are available through the “Help” button in some versions.
5 Languages Supported

• To change a language go to Configuration and select “Language”
• Choose the language you need from the drop down menu.
• Note that you will need to save the configuration, exit and re-enter the program for the language to register.
• Also note that only the software will be changed. All manuals are in English.

- English
- Spanish
- Italian
- French
- Portuguese
Contact Information

– Phone: (847) 940-9305
– Email:
  • support@flashcutcnc.com
  • sales@flashcutcnc.com
– Web: www.flashcutcnc.com