VERSION 9 CAD/CAM/CNC Tutorial
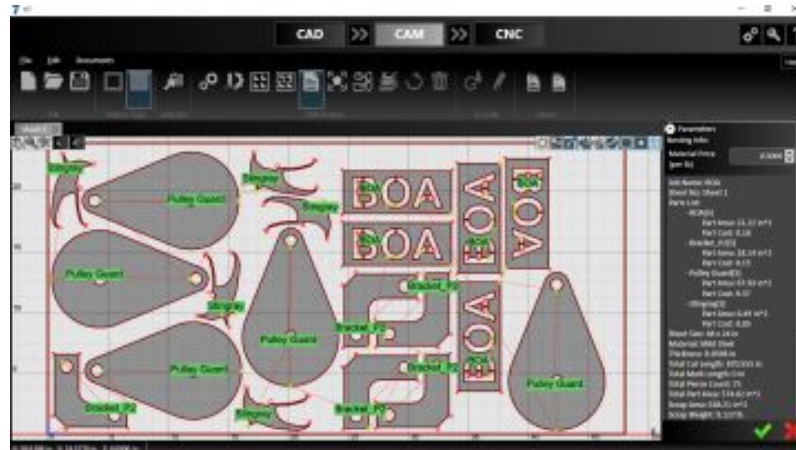
"After today I will be able to…"

…Draw a part in FlashCut CAD

…Use the Shape Library

…Import a color image

…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 CADCAM tools

…Design and cut pipe

…Use Smart360™ to produce holes

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 & 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 11 - Pipe Cutting CAD/CAM/CNC

Lesson 12 - Using Smart360^{TM}

Lesson 1:
Features, Installation, and Licensing

Application Features

Key:	
New or improved in V7	Feature
New or improved in V8	Feature
Included	Y
Optional A-la-carte	O
Not Included	N

Version 8 Feature List	Std	Pro
Laser Pointer Sfw Only	O	O
Laser Pointer Sfw and Hdw	O	O
Mill Router	O	O
Plasma	Y	Y
Oxyfuel	Y	Y
Waterjet	N	O
Laser	N	O
Marker and Plasma Marking	N	Y
Knife Cutting	N	O
DXF Export	O	Y
Part Numbers	N	O
Printing Support for Part Number Lables and Nest Report (Included with Part Numbers)	N	O
Advanced Text including True Type Font Support Without Exlploding and Curved Text Around Any Shape	N	Y
Part Shading	N	Y
3D Cutting Simulation	N	Y
Material Database (Cut Chart Support)	Y	Y
Customizable Cut Charts	Y	Y
Customizable Default Lead-in, Lead-out Strategies	Y	Y
Commonly Used Plate List	Y	Y
Multiple Simultaneous Fabheads	N	O
HPR Auto-Gas Control	N	O
Smart360	N	O
Pipe Cutting	N	O

Application

FLASHCUT CNC
CONTROL MADE SIMPLE

CAD Features

Key:	
New or improved in V7	Feature
New or improved in V8	Feature
Included	Y
Optional A-la-carte	O
Not Included	N

CAD		Std	Pro
	CAD	Y	Y
	Shape Library	O	Y
	Fix Drawing Tool	O	Y
	Bridge Entities Tool	O	Y
	Shape Welding Tool	N	Y
	Measuring Tool	Y	Y
	Part Shading	N	Y
	Free Form Stretching and Manipulation	Y	Y
	Layer Support	Y	Y
	Centerline Image Import	Y	Y
	Silhouette Image Import	Y	Y
	Multi Layer Color Image Import	N	Y
	Arc Fitting	N	Y
	DXF/DWG Import	Y	Y

FLASHCUT CNC
CONTROL MADE SIMPLE

CAM Features

	Std	Pro
CAM	Y	Y
TrueShape Nesting	O	Y
Automatic Kerf Compensation	Y	Y
Import of parts from external files	N	Y
Multi-Sheet and Fill Sheet TrueShape Nesting	O	Y
Grid Nesting	Y	Y
Corner Looping	N	Y
Nesting Report	N	O
Nesting Report + Part Numbers	N	O
Graphical editing of Leadin Lead out location	Y	Y
Tabbing	Y	Y
Multiple Tabs	N	Y
Overburn Support	Y	Y
Advanced Leadin Leadout Editing	N	Y
Animated 3D Cutting Simulation	N	Y
Assign different operations to each part	Y	Y
Cut Sequencing	Y	Y
Drag, Delete, Rotate, Copy parts on the nest	Y	Y
Operation Sequencing	Y	Y
Cutout or Part Mode	Y	Y
Cutom views of Kerf Width, Cut Direction , Toolpath, Rapids and Part Geometry	Y	Y
Customizable Post Processor	Y	Y
Collision Avoidance	N	O
Advanced Small Hole Processing	N	Y

CAM

Key:

New or improved in V7	Feature
New or improved in V8	Feature
Included	Y
Optional A-la-carte	O
Not Included	N

FLASHCUT CNC
CONTROL MADE SIMPLE

CNC Features

Key:	
New or improved in V7	Feature
New or improved in V8	Feature
Included	Y
Optional A-la-carte	O
Not Included	N

	Std	Pro
CNC	Y	Y
THC Sfw and Hdw	O	O
Plasma Communications Sfw and Hdw (RS422 or RS485)	O	O
Advanced G-Code File Management	N	O
Automatic Lead-In on Restart	N	Y
Dyamic Ventilation	N	O
Dynamic THC On/Off	N	Y
Jump to Line	Y	Y
Dry Run	Y	Y
Momentary Run	Y	Y
Run File Forward and Backwards	Y	Y
Motor Current Sensing	N	O
Workpiece Alignment	O	O
Rotary Axis Support	N	O
Kerf Crossing Detection	Y	Y
Progress Meter	N	Y
Move to Point in Viewport	N	Y
Rip Cutting	O	Y
THC Performance History Chart	Y	Y
Auto Tool Change Support	N	O
Camera Vision Support for Fiducial or Registration Marks	N	O
Smart Touch Off	N	Y

CNC

FLASHCUT CNC
CONTROL MADE SIMPLE

Installation

1. Launch the FlashCut CNC Version 9 installer
2. Choose *Run* in the following
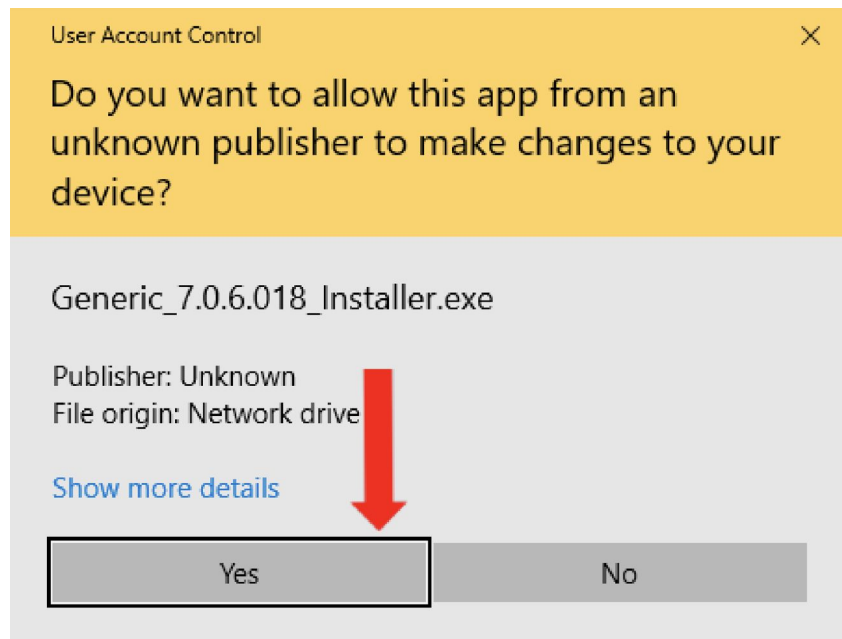
Installation
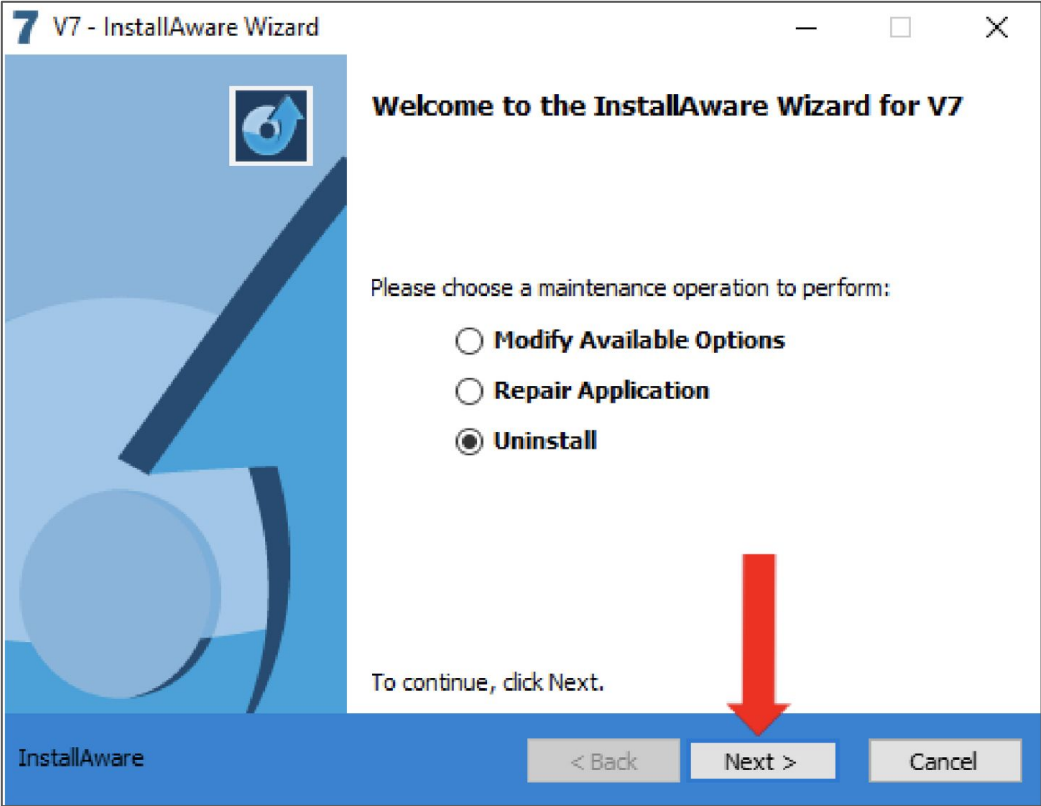
- If you have a previous version of V9, it will be automatically uninstalled.
- If you already have V5-V8, it will remain installed.

Installation
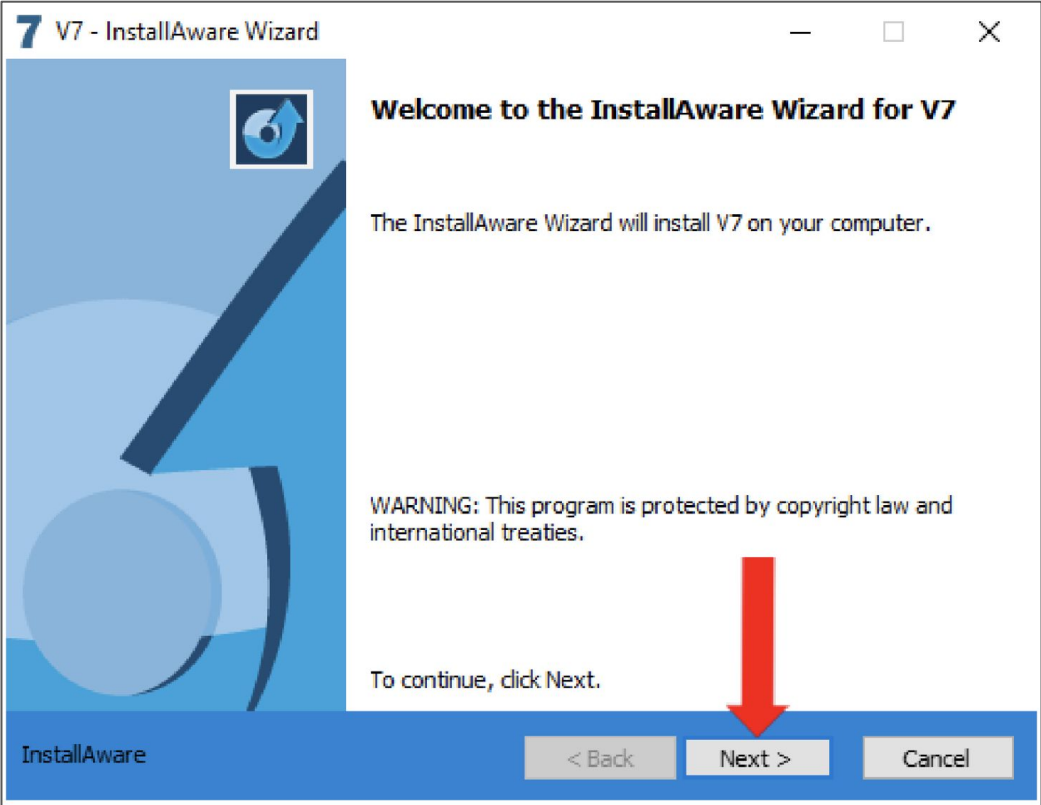
- The installer will launch after any previous version is successfully uninstalled.

Installation
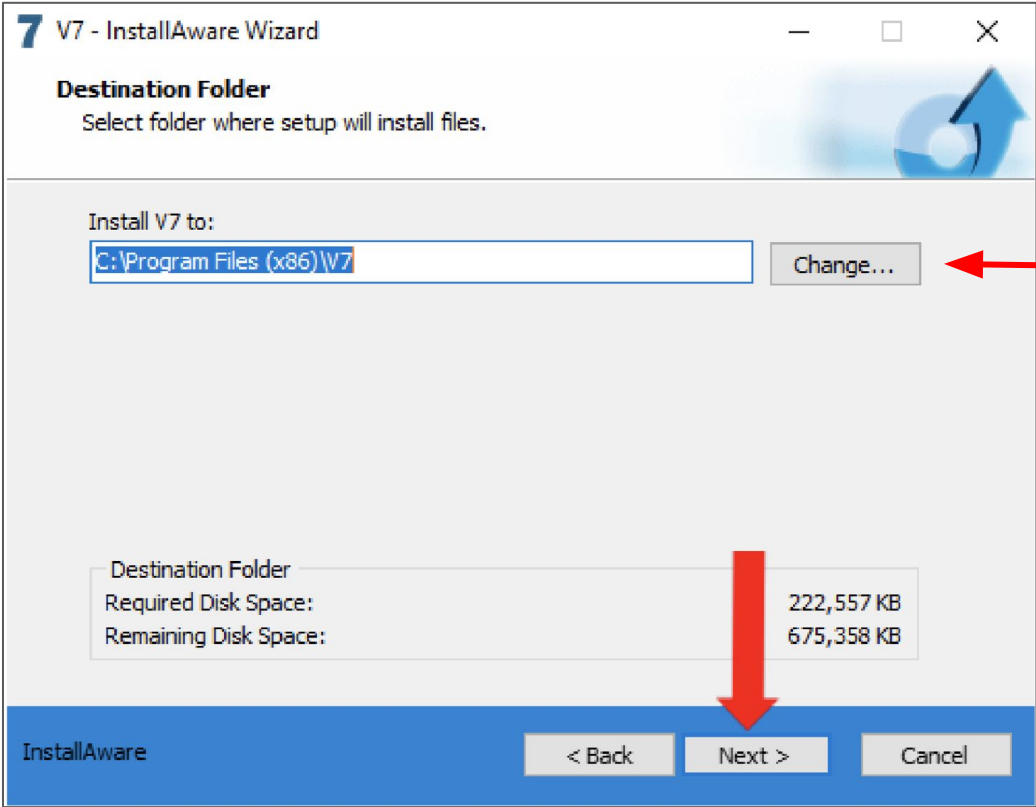
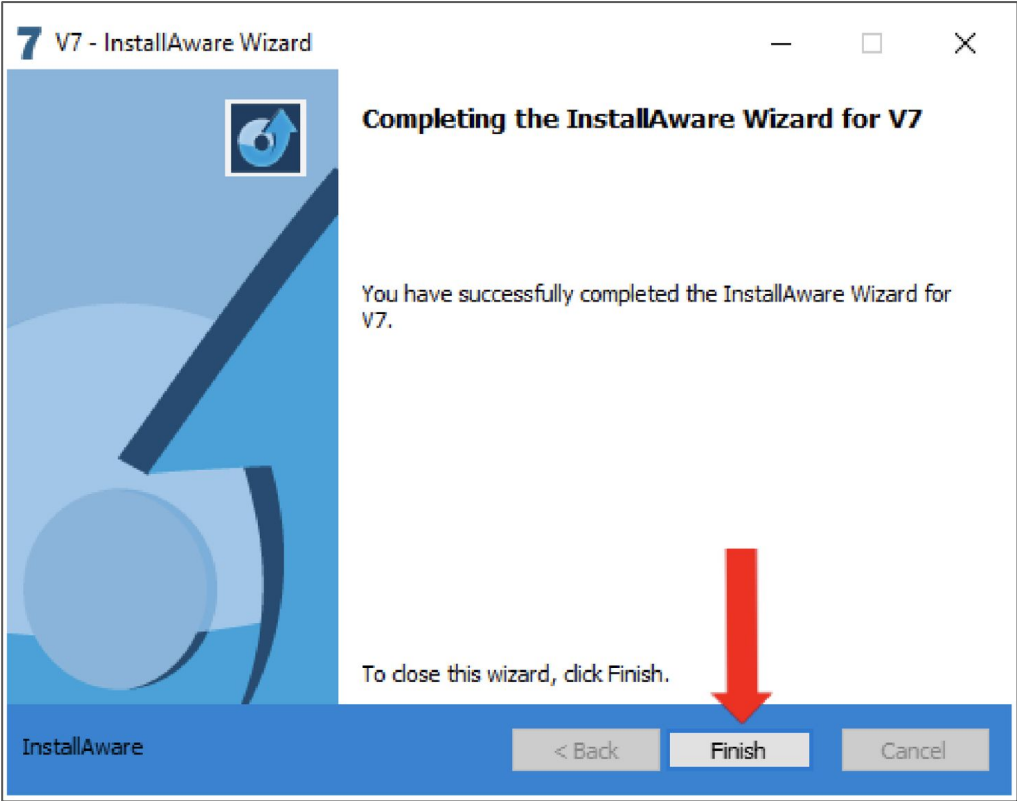- Choose the folder location for the software to install.

Installation
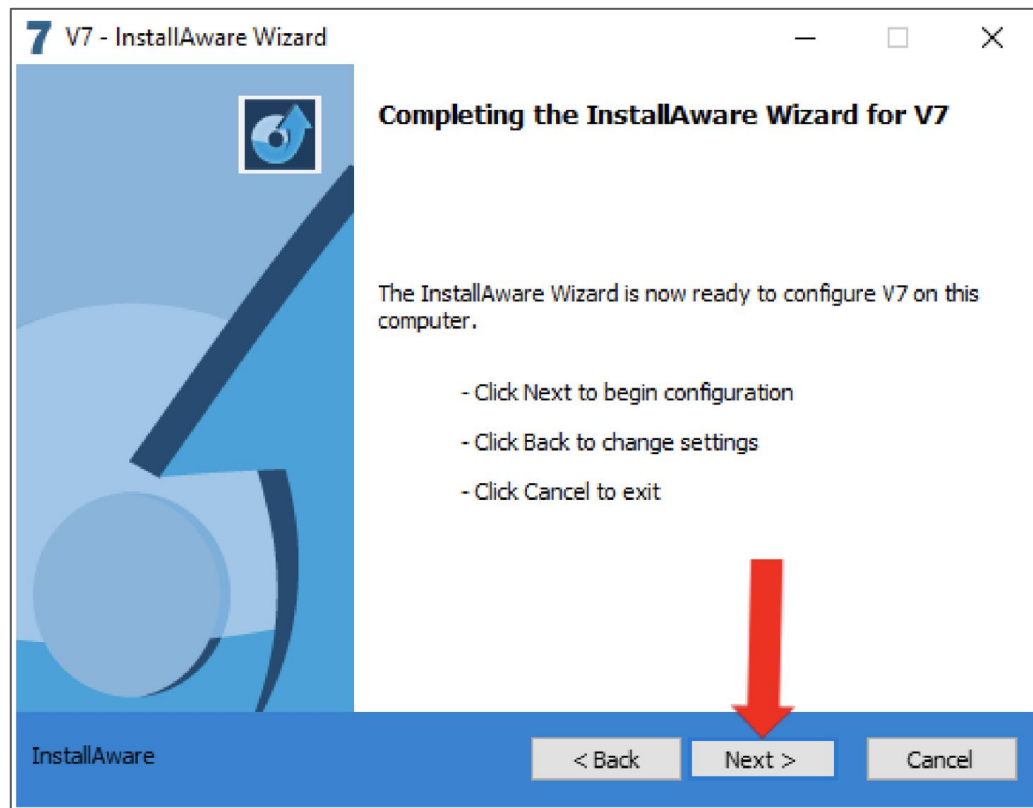
- Confirm the chosen settings.

Installation
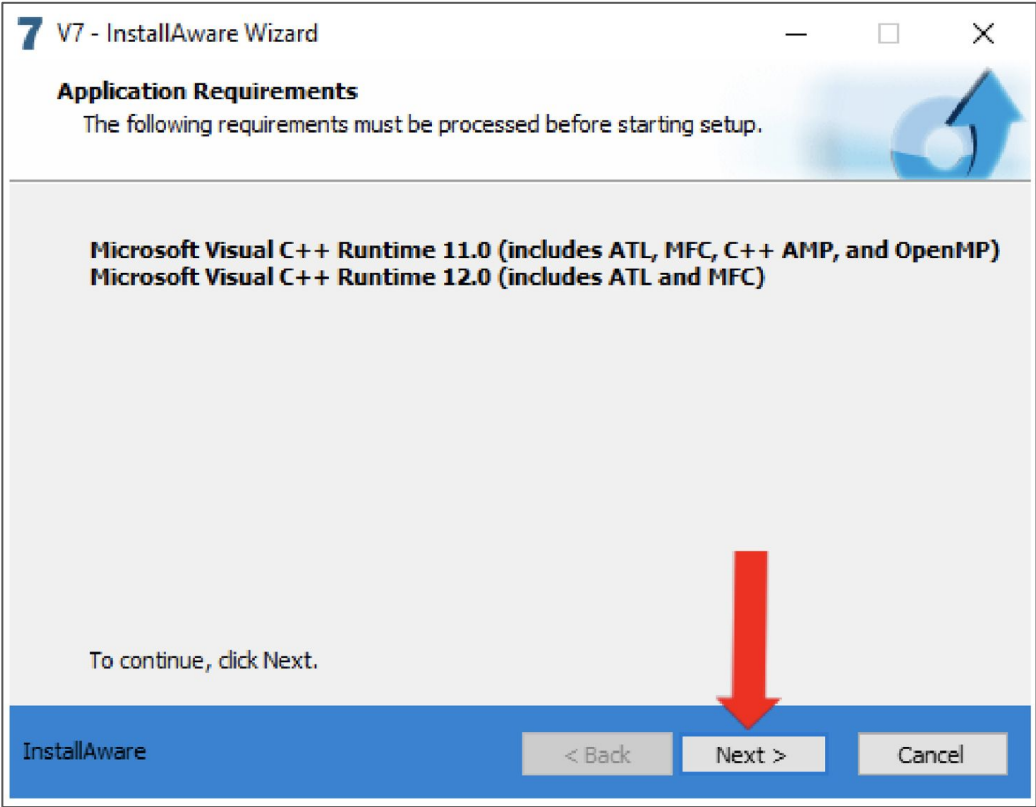
- Automatically install required applications.

Installation

- The application is successfully installed.

Licensing

1. Each system comes with two software licenses:
   - One license embedded in the controller for any PC that connects
   - 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"; this 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 window

Then, locate the appropriate license in the pop-up menu.

First, click the key icon.

CAD >> CAM >> CNC

The system may be licensed by a controller license or PC license. Both types are available for any licensed system.

Controller License
- Resides on the controller
- The software is licensed when connected to a licensed controller

Import License File...

PC License
- Resides on the PC
- The software is always licensed, regardless of connection to a controller

Deactivate PC License...

APPLICATION
+ Advanced Text Support
+ CAD/CAM for Milling & Routing
+ Cutting Simulation
+ Display Part Numbers
+ DXF Export

APPLICATION
+ Advanced Text Support
+ CAD/CAM for Milling & Routing
+ Cutting Simulation
+ Display Part Numbers
+ DXF Export

FLASHCUT CNC
CONTROL MADE SIMPLE

Activating PC License

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. Press the Activate PC License button

7. Copy the License Key from the licensing email and paste it in the License Key text box by typing or using the Paste From Clipboard option

8. Click the License Software button
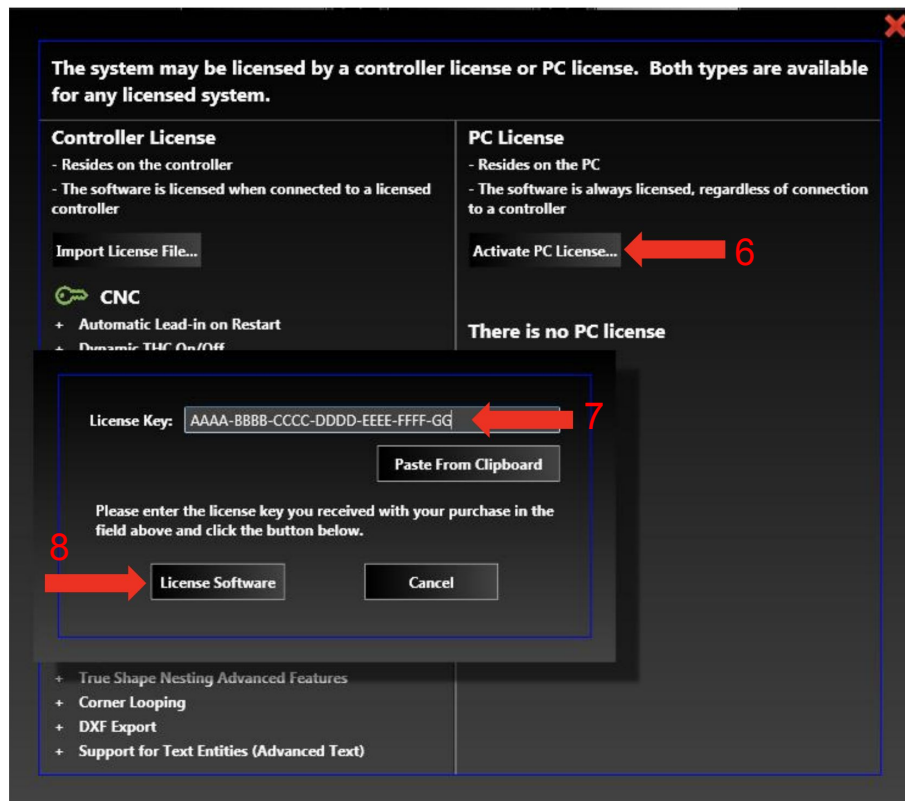
Activating PC License

● 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 same steps as above

● <u>You must have an internet connection on</u> <u>both PCs for this to work</u>

The system may be licensed by a controller license or PC license. Both types are available for any licensed system.

Controller License
- Resides on the controller
- The software is licensed when connected to a licensed controller

Import License File...

🔑 **CNC**
+ Automatic Lead-in on Restart
+ Dynamic THC On/Off
+ Dynamic Ventilation
+ THC
+ Laser Pointer
+ Plasma Communications
+ Advanced G-Code File Management
🔑 **CAD**
+ Shape Tool
+ DXF Export
+ Fix Drawing Tool
+ Bridge Entities Tool
🔑 **CAM**
+ Import Parts from External Files
+ True Shape Nesting Advanced Features
+ Corner Looping
+ DXF Export
+ Support for Text Entities (Advanced Text)

PC License
- Resides on the PC
- The software is always licensed, regardless of connection to a controller

Deactivate PC License...

🔑 **CNC**
+ Automatic Lead-in on Restart
+ Dynamic THC On/Off
+ Dynamic Ventilation
+ THC
+ Laser Pointer
+ Plasma Communications
+ Advanced G-Code File Management
+ Motor Current Sensing
🔑 **CAD**
+ Shape Tool
+ DXF Export
+ Fix Drawing Tool
+ Bridge Entities Tool
+ Weld Entities Tool
+ Curve Text Tool (Advanced Text)
🔑 **CAM**
+ Import Parts from External Files
+ True Shape Nesting
+ True Shape Nesting Advanced Features
+ Corner Looping
+ DXF Export
+ Support for Text Entities (Advanced Text)

FLASHCUT CNC
CONTROL MADE SIMPLE

Activating PC License

- Once you have activated your license, you can now view the activated features listed on the license panel

- The green key indicates activated features, and the grey lock indicates features in evaluation mode
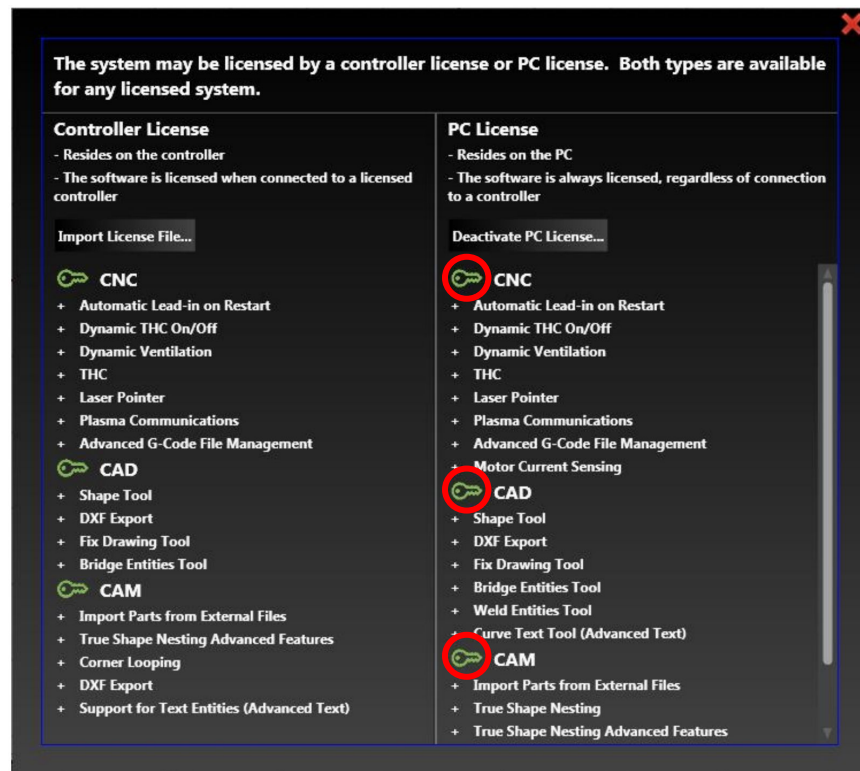
Activating Controller License

1. To activate the controller license, you must do the following:
   a. Make sure your USB cable is connected and your controller is turned on
   b. Go to the CNC tab
   c. Hit the *Connect* button – which will turn green when connected successfully

2. Once you are connected to the controller, it is only necessary to do a "software only" license if you want to license a second PC or if the controller PC will not always be connected to the controller

Lesson 2:
Drawing in CAD
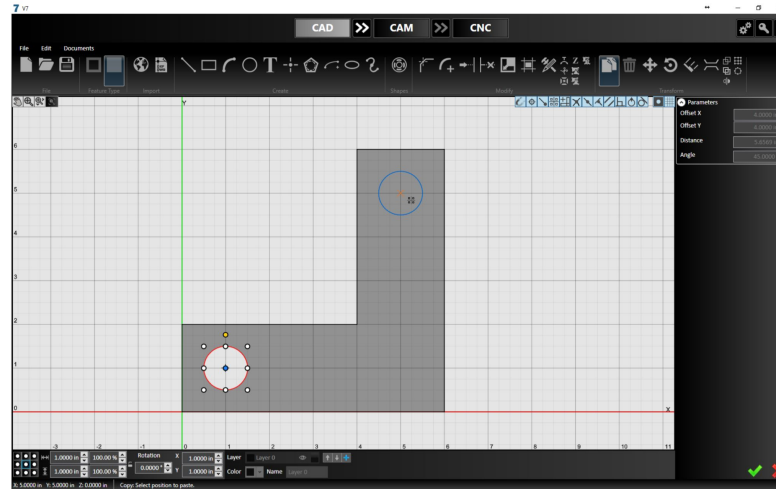
Create Tools

Create Tools circled on toolbar below
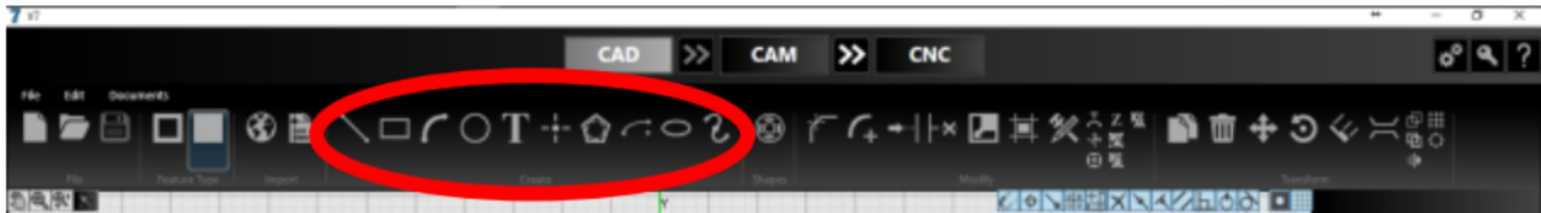
"Create" section labeled below.

Line Rectangle Arc Circle Text Point Polygon Elliptical Arc Ellipse Spline

Create

Shape Library

PRO

Shape Library circled on toolbar below
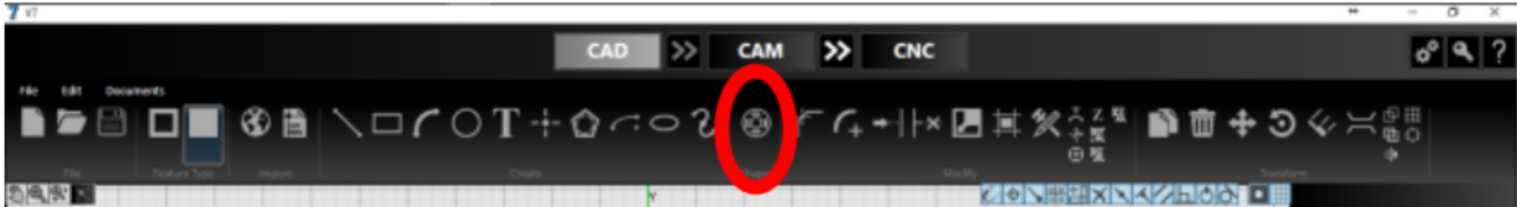
The Shape Library provides premade shapes you can manipulate and add to your drawing field

Shapes

FLASHCUT CNC
CONTROL MADE SIMPLE
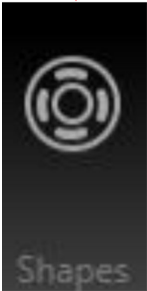
Modify Tools

Modify Tools circled on toolbar below
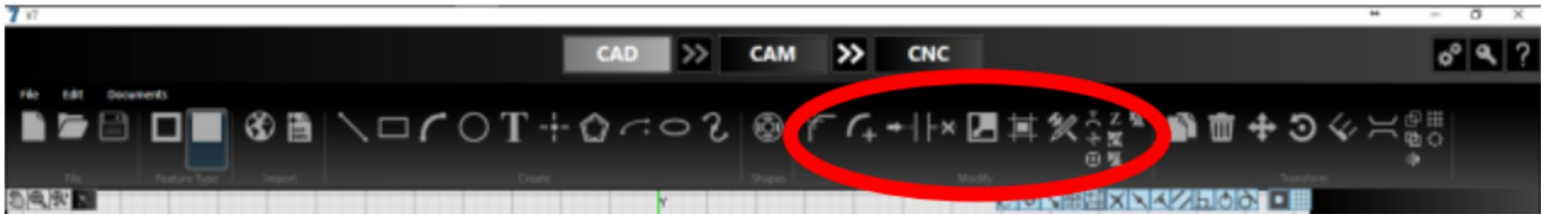
"Modify" section labelled below

Corner Treatment | Extend | Trim | Scale | Offset
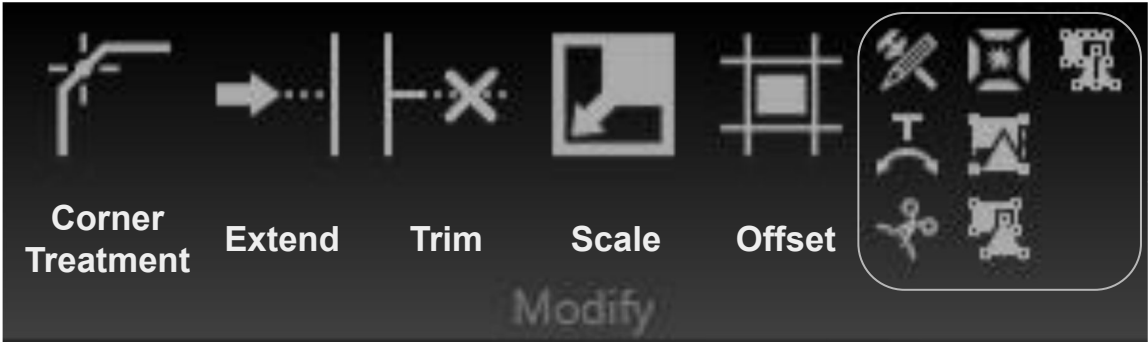
- Fix Drawing *PRO*
- Curved Text *PRO*
- Cut
- Explode
- Group
- Ungroup
- Ungroup All

FLASHCUT CNC
CONTROL MADE SIMPLE

Transform Tools

Transform Tools circled on toolbar below
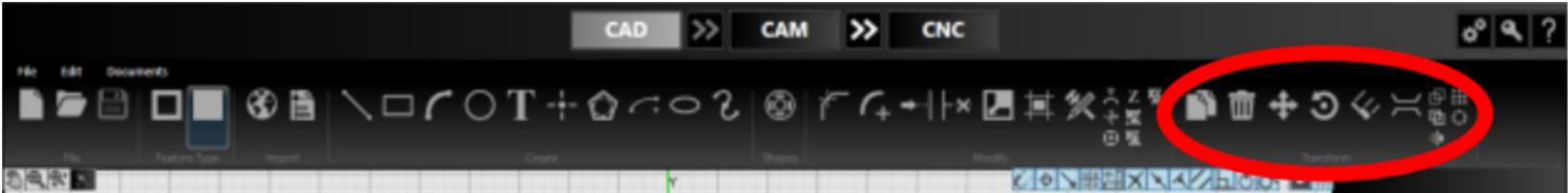
"Transform" section labelled below

Copy   Delete   Move   Rotate   Measure   Bridge

Transform
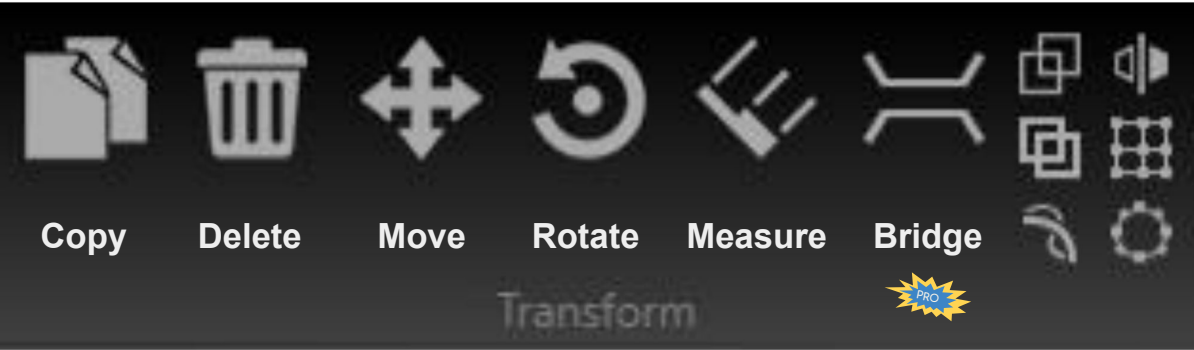
- Boolean Operations PRO
- Shape Welding PRO
- Arc Fitting PRO
- Mirror
- Linear Matrix
- Circular Matrix

FLASHCUT CNC
CONTROL MADE SIMPLE

Toggle Snap and Palate Tools

Snap Tools:

- All Snaps Off
- Center
- Endpoint
- Grid
- Horizontal/ Vertical
- Intersection

- Midpoint
- Nearest
- Parallel
- Perpendicular
- Quadrant
- Tangent

Palate Tools:

- Part Shading On/Off *PRO*
- Palate Grid On/Off

FLASHCUT CNC
CONTROL MADE SIMPLE

Importing Files

Bitmaps – BMP, DIB, JPG, JPEG, PNG, PBM, PGM, PPM, SR, RAS, TIFF, and TIF
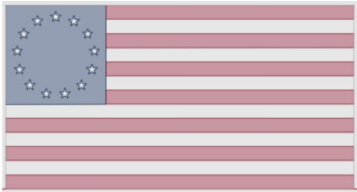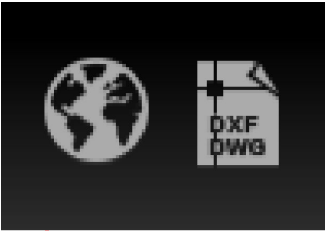
Color Image PRO

Centerline

Silhouette

Import Silhouette

Import Centerline

Import Color Image

DXF/DWG Drawings

FLASHCUT CNC
CONTROL MADE SIMPLE
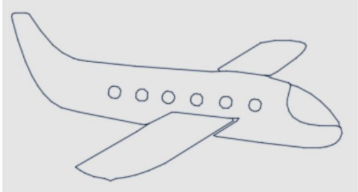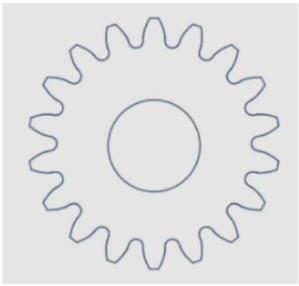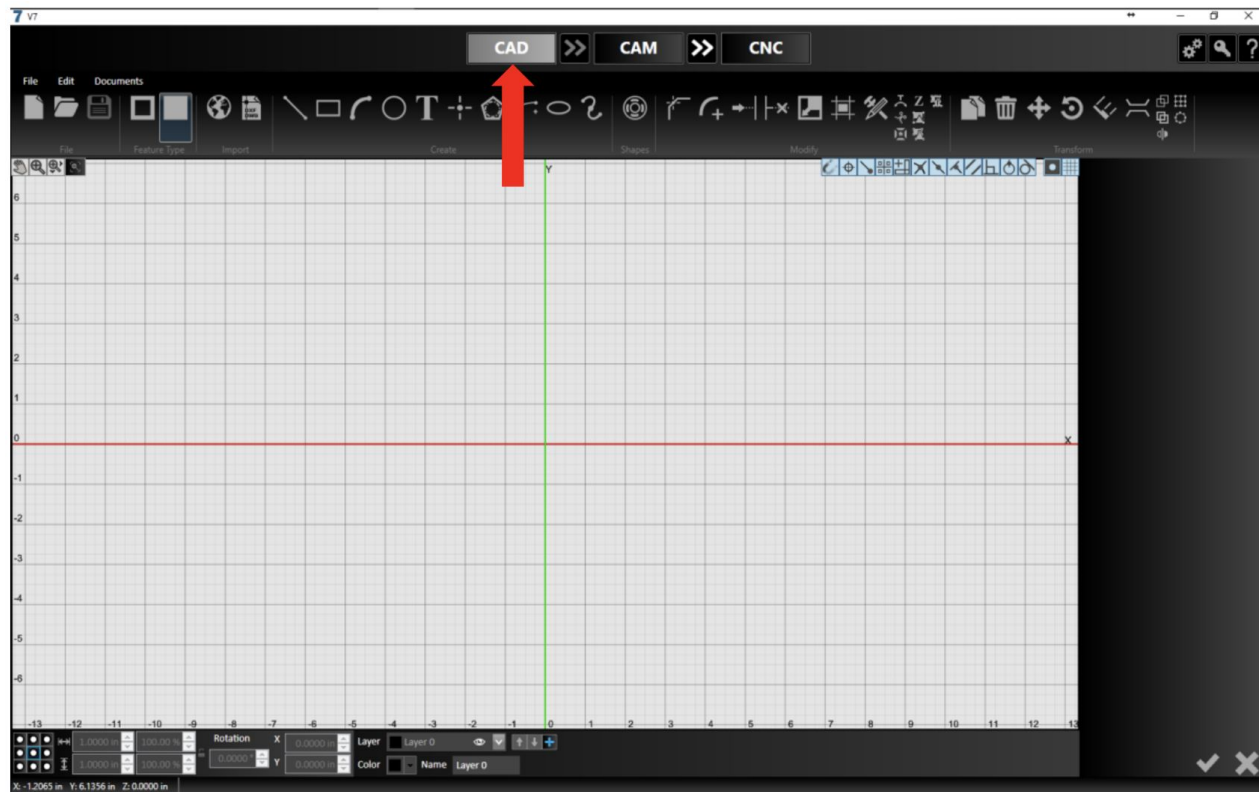
Making a Bracket

Open a new **2D file** and start by clicking on the *CAD* tab to begin your drawing

Making a Bracket
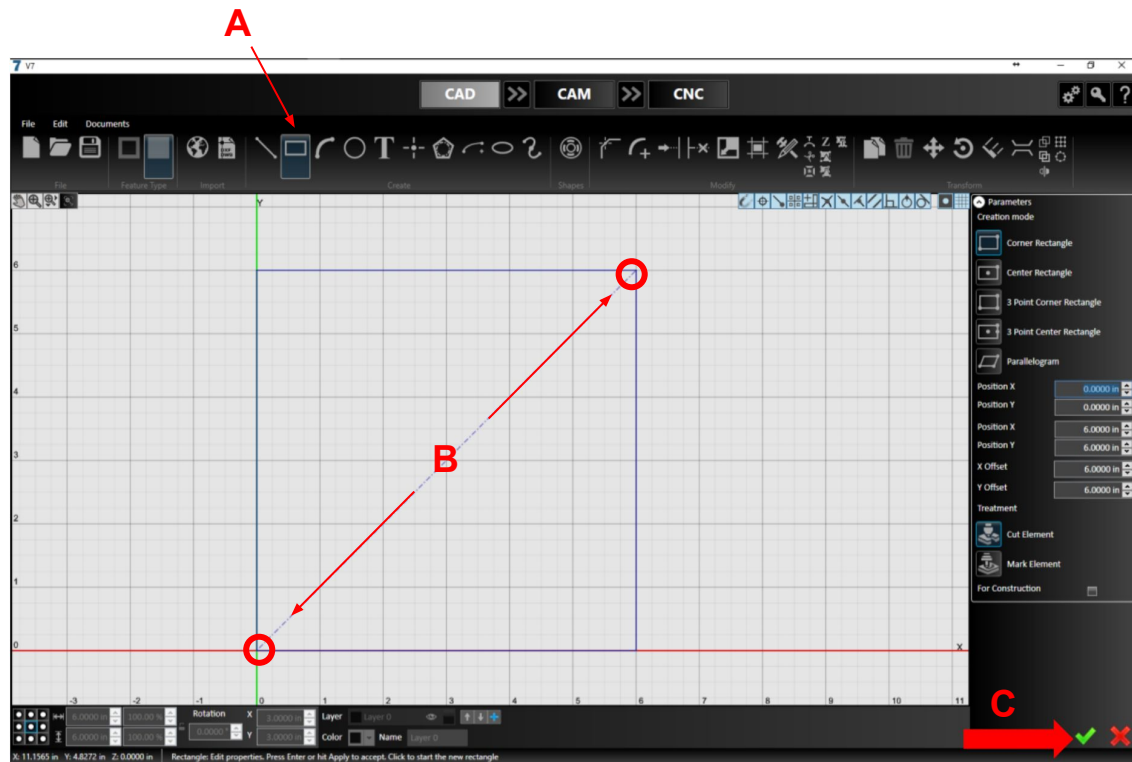
Step 1: Make a 6" by 6" Square.

A. Choose the *Rectangle* tool
B. Define the dimensions and placement by either clicking opposite corners with your mouse <u>or</u> setting the XY coordinates in the parameters panel
  a. Make sure the <u>lower left corner</u> of the square is <u>located at (0,0)</u>
C. Click the green check at the bottom of the parameters panel to "Check It In"

Making a Bracket
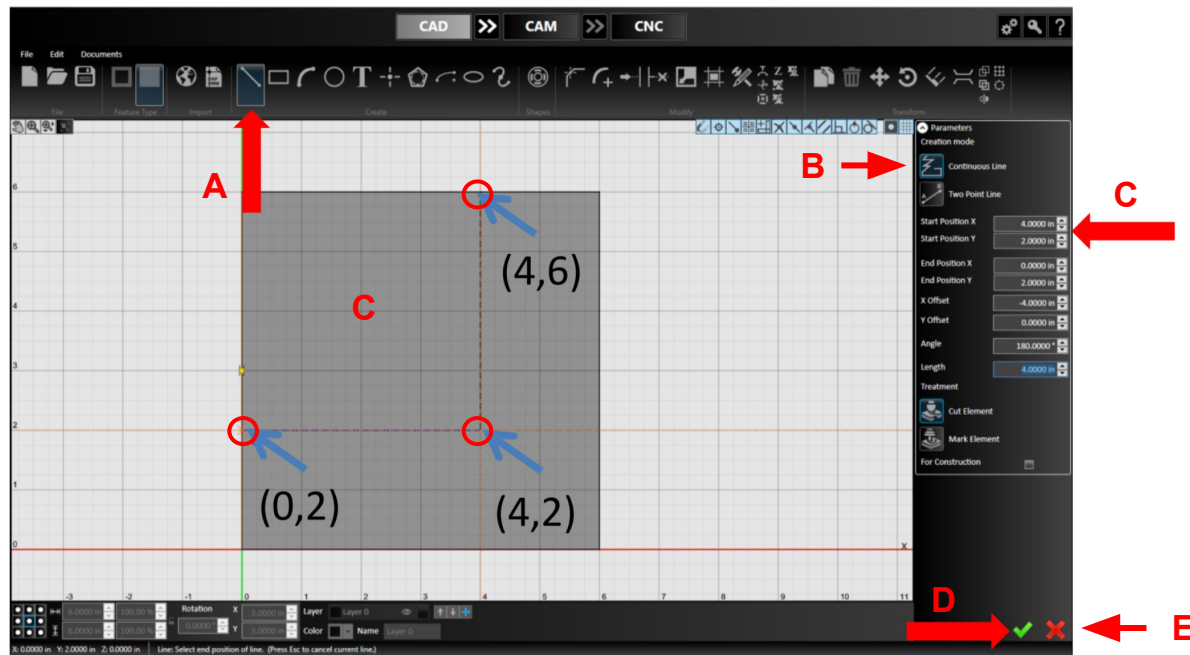
Step 2: Make a Polyline.

A. Choose the *Line* tool in the toolbar
B. Select *Continuous Line* in the parameters panel
C. Select the following three points with your mouse <u>or</u> by typing the XY coordinates: (4,6), (4,2), (0,2).
D. Click the green check to confirm your drawing
E. To end the polyline, click the red X or the *Esc* key

Making a Bracket
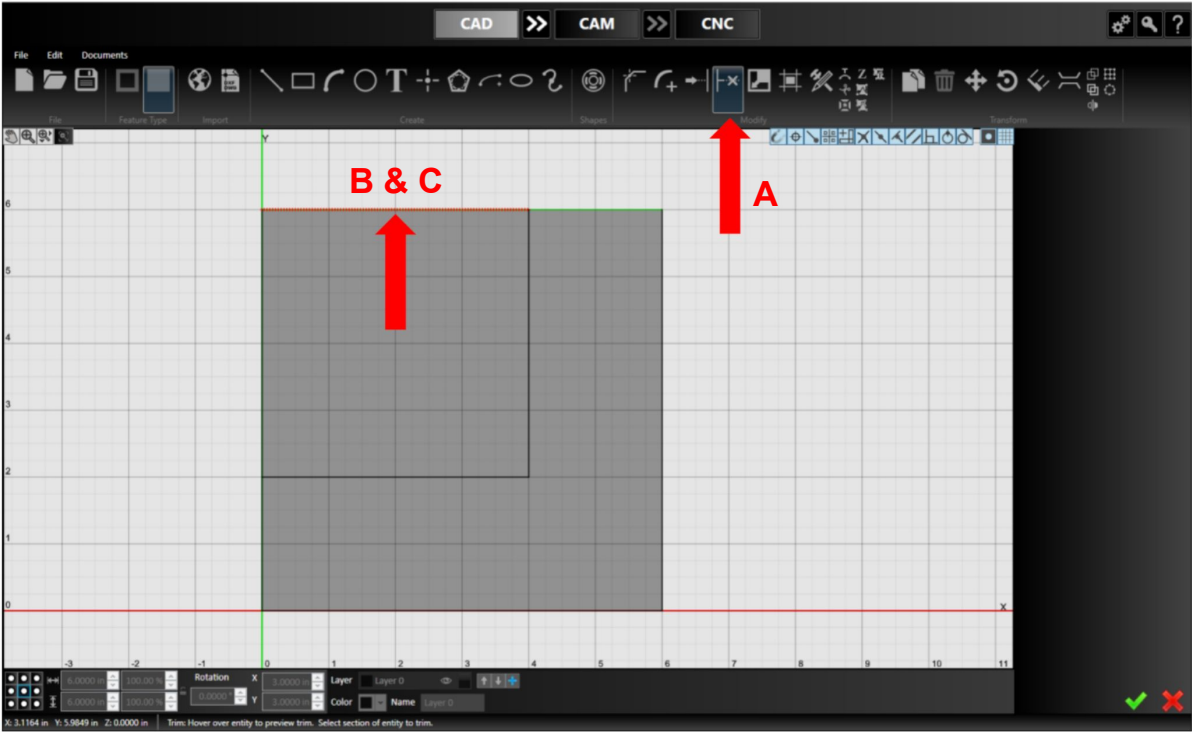
Step 3: Trim the Excess.

A. Select the *Trim* tool
B. Hover your mouse over the line segment you want to trim; it will turn red
   a. Note: the portion of the line segment that you will keep will turn green
C. Click the red segment to remove

Making a Bracket
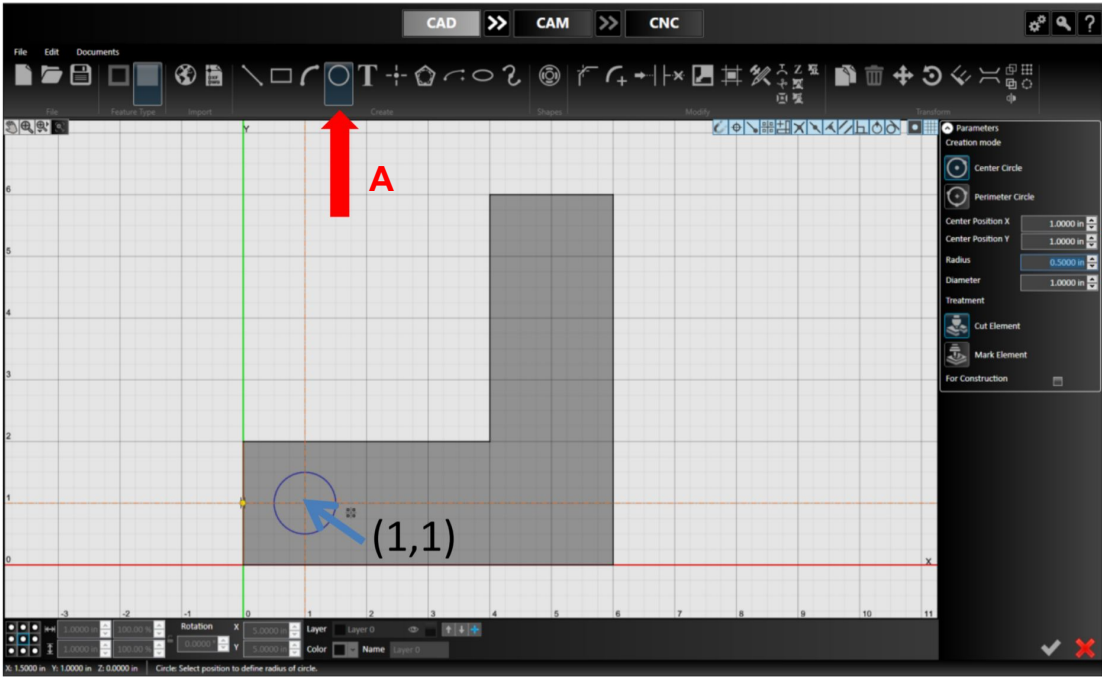
Step 4: Draw a Circle.

A. Select the *Circle* tool
B. Select the center point (1,1) with your mouse <u>or</u> type the XY coordinates in the parameter panel
C. Set the diameter to 1 with your mouse <u>or</u> enter the value into the parameter panel

Making a Bracket
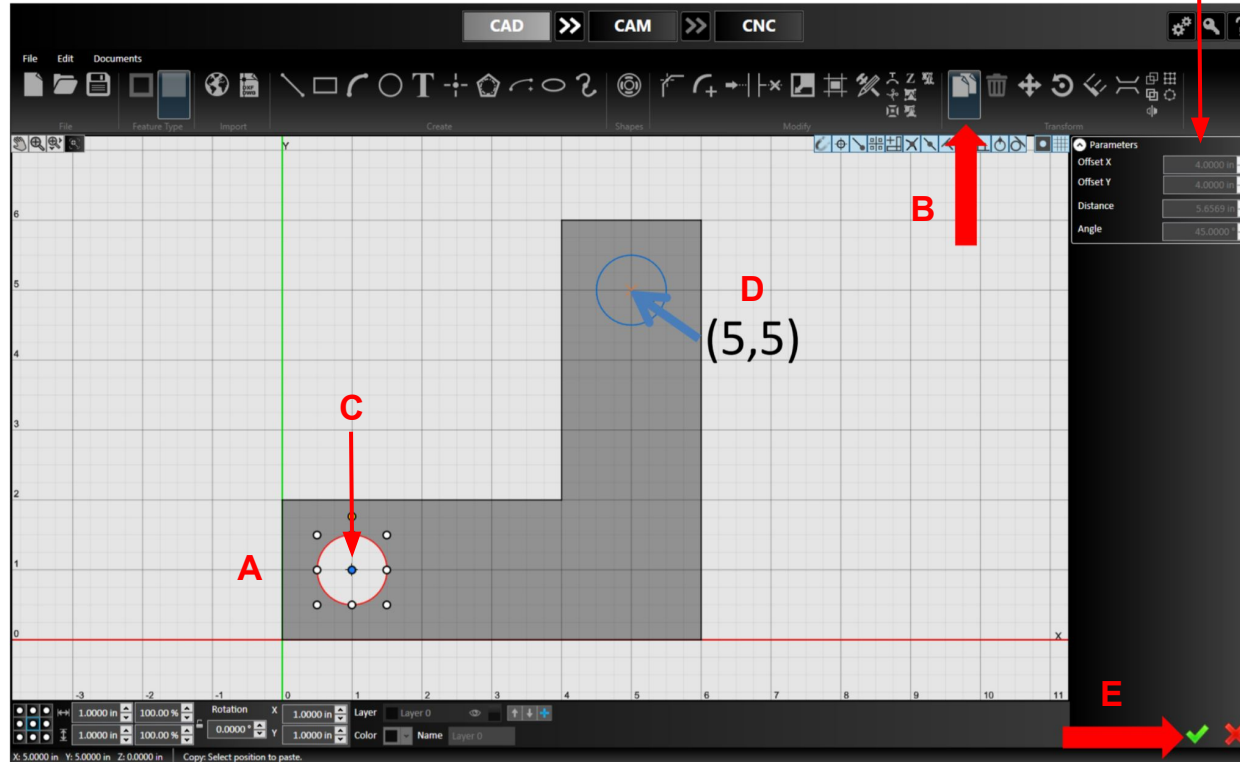
Step 5: Copy the Circle.

A. Select the existing circle; it will turn red
B. Select the *Copy* tool
C. Select the center of the existing circle as a reference point
D. Select the new center point at (5,5) with your mouse or enter (4,4) for the Offset in the parameters panel
E. Check it in

Making a Bracket
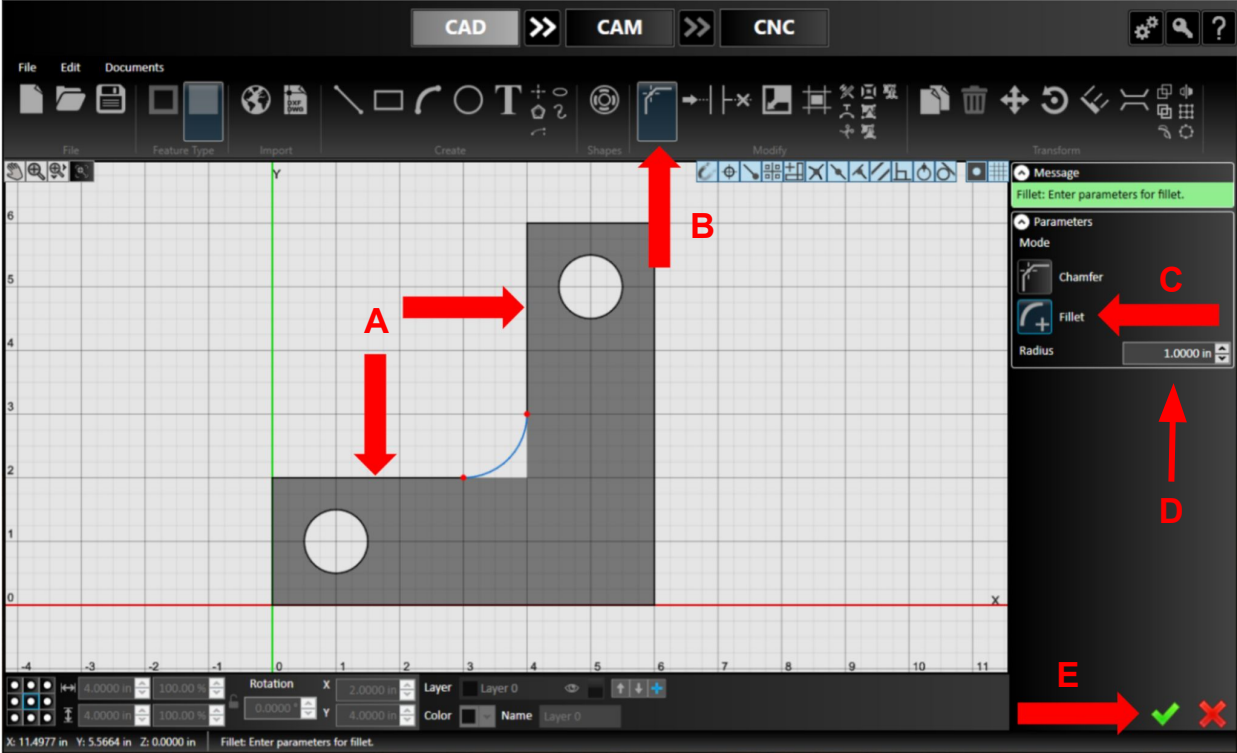
Step 6: Create a Fillet.

A. Select the inner corner or two adjacent lines to fillet; they will turn red

B. Choose the *Corner Treatment* tool in the toolbar

C. Select *Fillet* in the parameters panel

D. Adjust the radius of the edge with your mouse <u>or</u> enter a radius of 1.0 in the parameters panel

E. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE

Making a Bracket
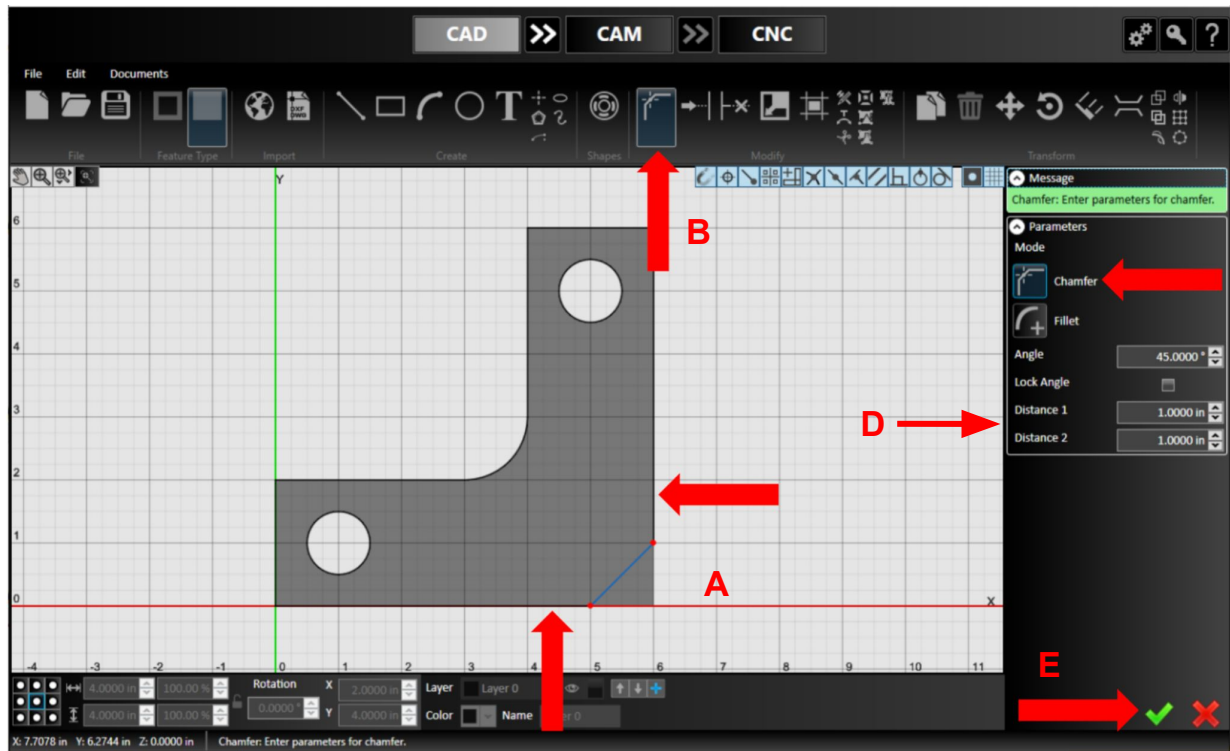
Step 7: Create a Chamfer.

A. Select the lower right corner or the two adjacent lines to chamfer; they will turn red

B. Choose the *Corner Treatment* tool again

C. Select *Chamfer* in the parameters panel

D. Either drag to size or enter a length of 1.0 for Distance 1 & 2

E. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE
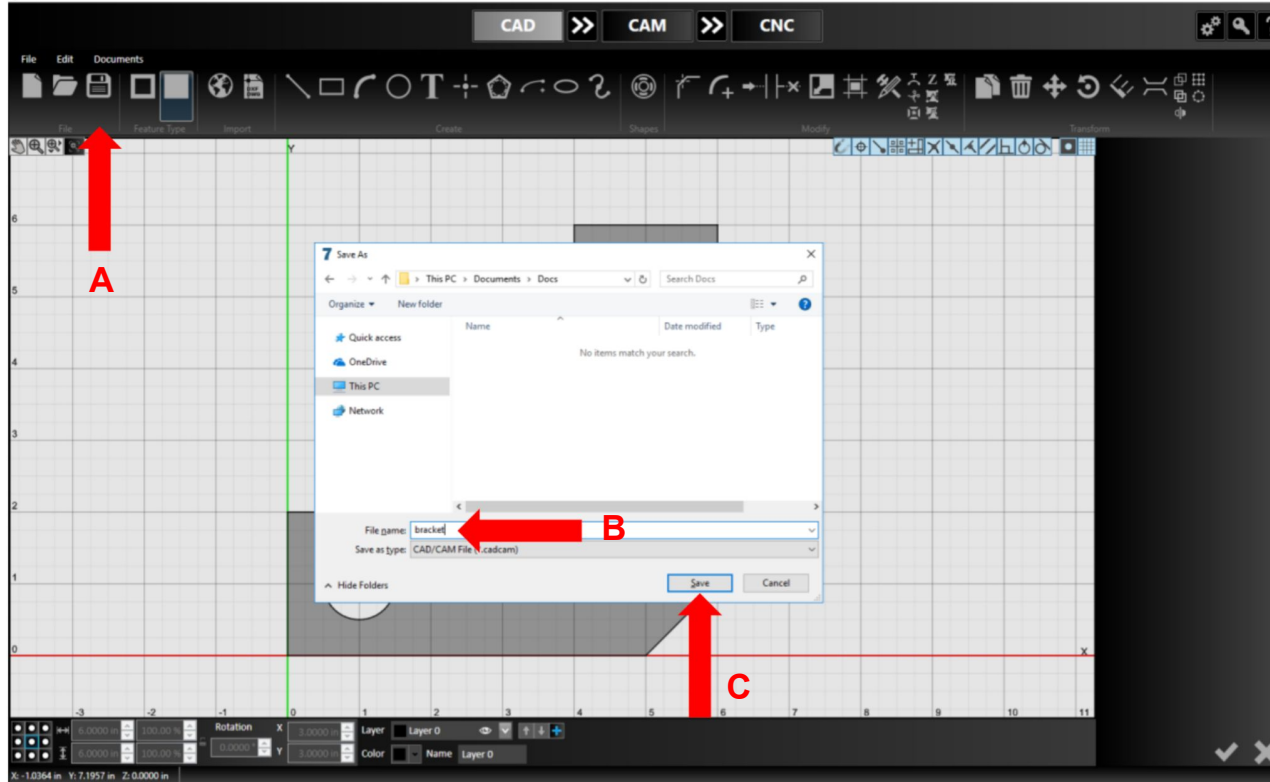
Making a Bracket

Step 8: Save the Drawing.

A. Select the *Save* icon <u>or</u> from the *File* menu choose *Save*

B. Name the file "Bracket" when prompted to do so

C. Hit the *Save* button

Lesson 3:
Making a Toolpath in CAM

Using CAM: Selection Tool

- Used to select groups of entities:
  - Breaks – All; Perimeter; Cutouts
  - Operations – All; Perimeter; Cutouts; Open Profiles; Marks; Center Pierces; Drilling
  - Geometry – All; Perimeter; Cutouts; Holes; Other
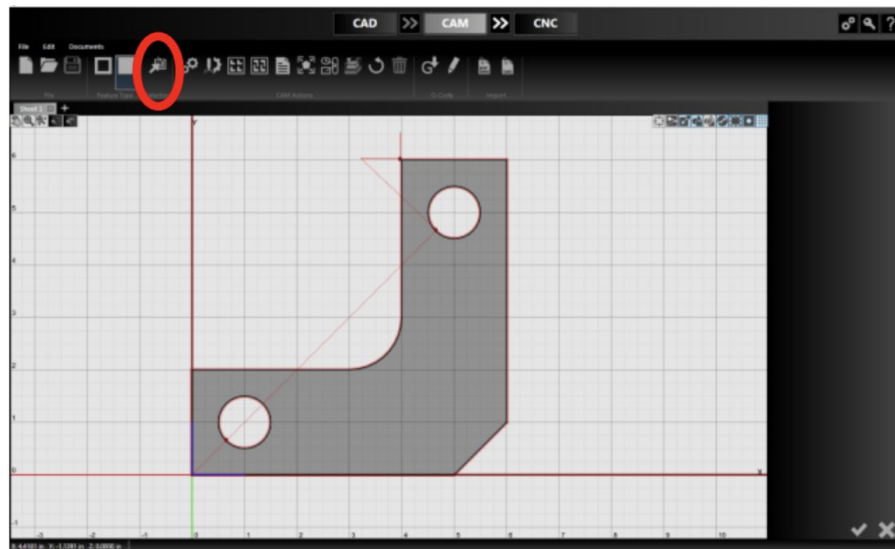
✳ Ctrl-click will also add or subtract entities from the current selection

FLASHCUT CNC
CONTROL MADE SIMPLE

Using CAM: Display Options
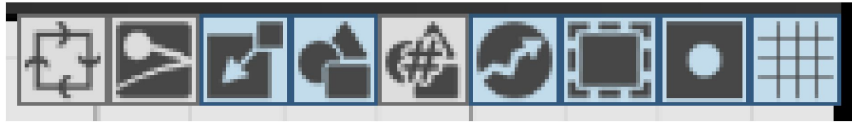
Convenient way of displaying and hiding:

- *Cutting Direction*
- *Kerf Compensation*
- *Perimeter of Material*
- *Parts*
- *Part Numbers* OPT
- *Rapid Moves*
- *Toolpath*
- *Part Shading* PRO
- *Gridlines*

FLASHCUT CNC
CONTROL MADE SIMPLE
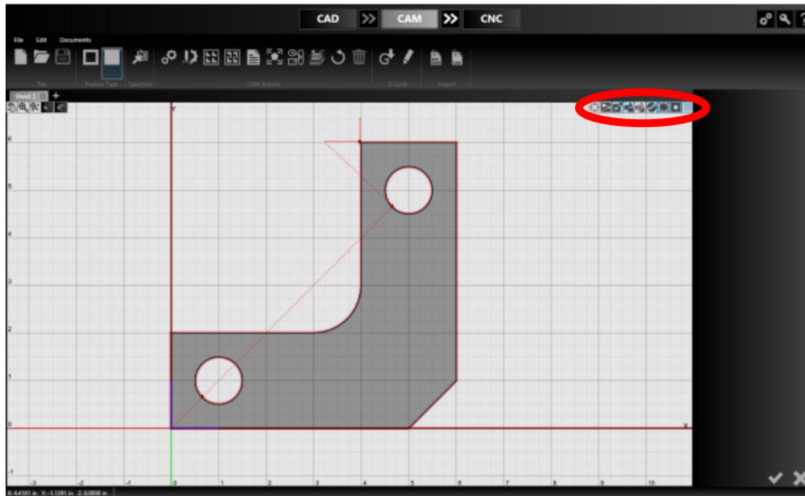
Using CAM: G-Code Generation
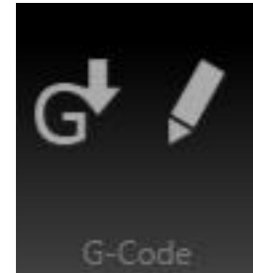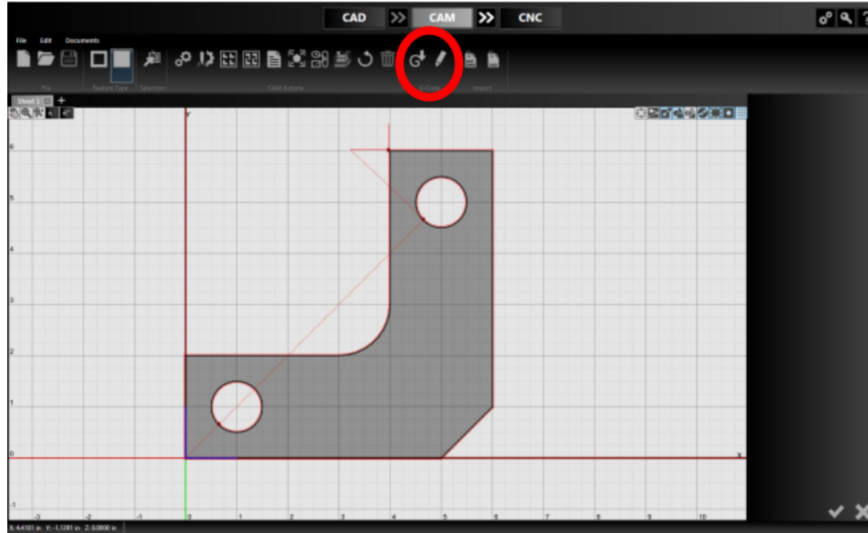
Use to *Generate* and *Edit* G-Code

Using CAM: CAM Actions
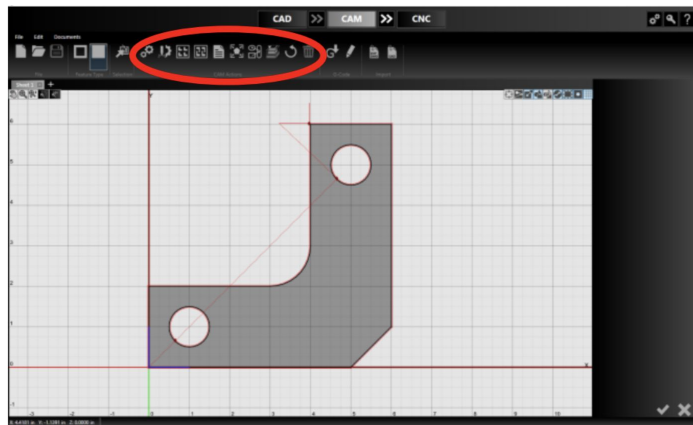
- *Project Settings*
- *Fab Head (Plasma) Settings*
- *Break Settings*
- *Tab Settings* **PRO**
- *Grid Nesting*
- *True-Shape Nesting* **PRO**
- *Nesting Report* **OPT**
- *Cutting Order*
- *Cutting Simulation* **PRO**
- *Reset Tool Path*

Using CAM: View Tools

- *Sheet Number*
- *Pan*
- *Zoom*
- *Zoom Window*
- *Zoom Workpiece Extents*
- *Zoom Toolpath Extents*
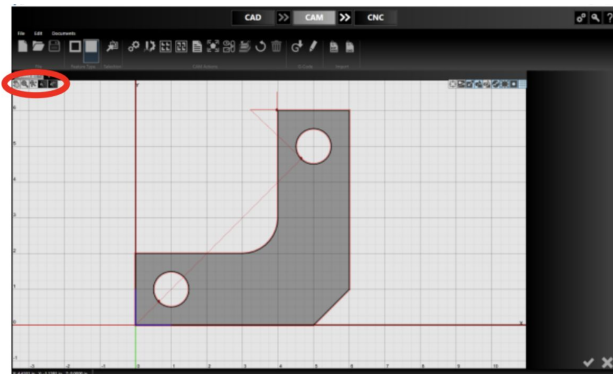
Making a Toolpath in CAM
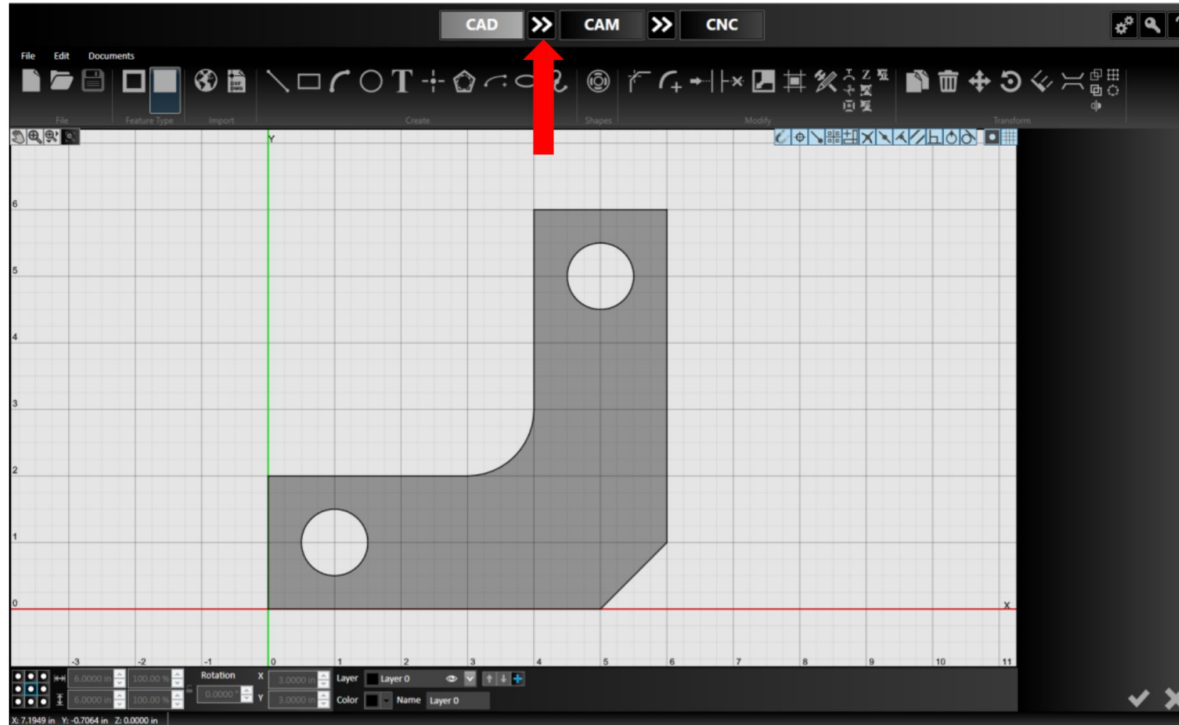
Step 1: Transfer CAD Bracket Drawing to CAM.

A. Hit the double arrow button between *CAD* and *CAM*

   a. An initial toolpath will be generated with your default parameters

Making a Toolpath in CAM

A

B

C

D

E & F

G

Lead-In Head

Breakpoint
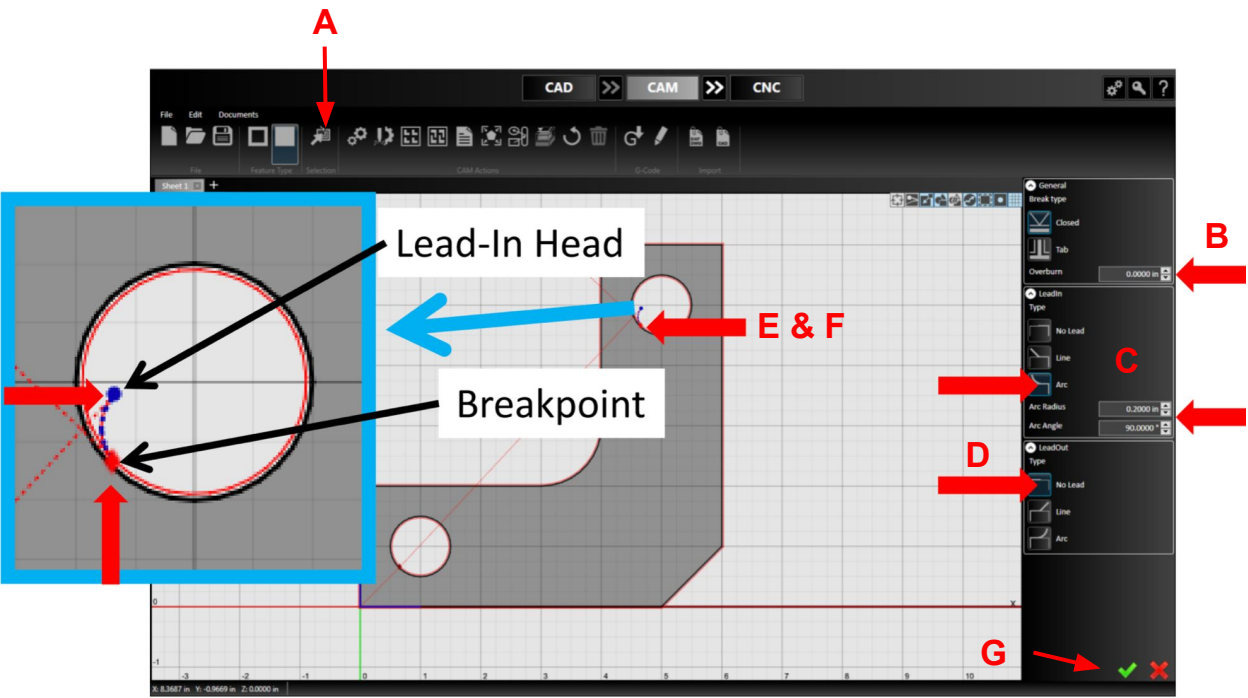
Step 2: Setting Lead-Ins and Lead-Outs.

A. Click the *Selection* button; a drop down menu will appear
   a. Hover over the *Breaks* option and select *Cutout Breaks*
B. On the parameters panel, find the *General* section, and set *overburn* to 0
C. In the *Lead-In* section, specify the following:
   a. *Type: Arc*
   b. *Radius*: 0.125
   c. *Angle*: 120
D. In the *Lead-Out* section, select *no lead-out*
E. With mouse, move lead-in to desired location by dragging the breakpoint
F. Dynamically adjust parameters by dragging the lead-in head
G. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE

Making a Toolpath in CAM

Sheet Outline

Step 3: Setting Workpiece Dimensions.

A. Click the *Zoom to Workpiece Extents* button

B. Select *Project Settings* icon in the *CAM Actions* section on toolbar

C. In the parameters panel, make sure *Default Cutting Head* is correct

    a. **Check it in**

D. Select sheet outline; it will turn green

E. In the parameters panel, Set workpiece dimensions to 48" wide x 24" long

F. Set material type and properties

G. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE

Making a Toolpath in CAM

Step 4: Grid Nesting.

A. Select the *Grid Nesting* button in the *CAM Actions* section

B. Set *Count* to 18

C. Set *Spacing X/Y* to 1.0

D. Click *Start Nesting*

E. Check it in

Making a Toolpath in CAM
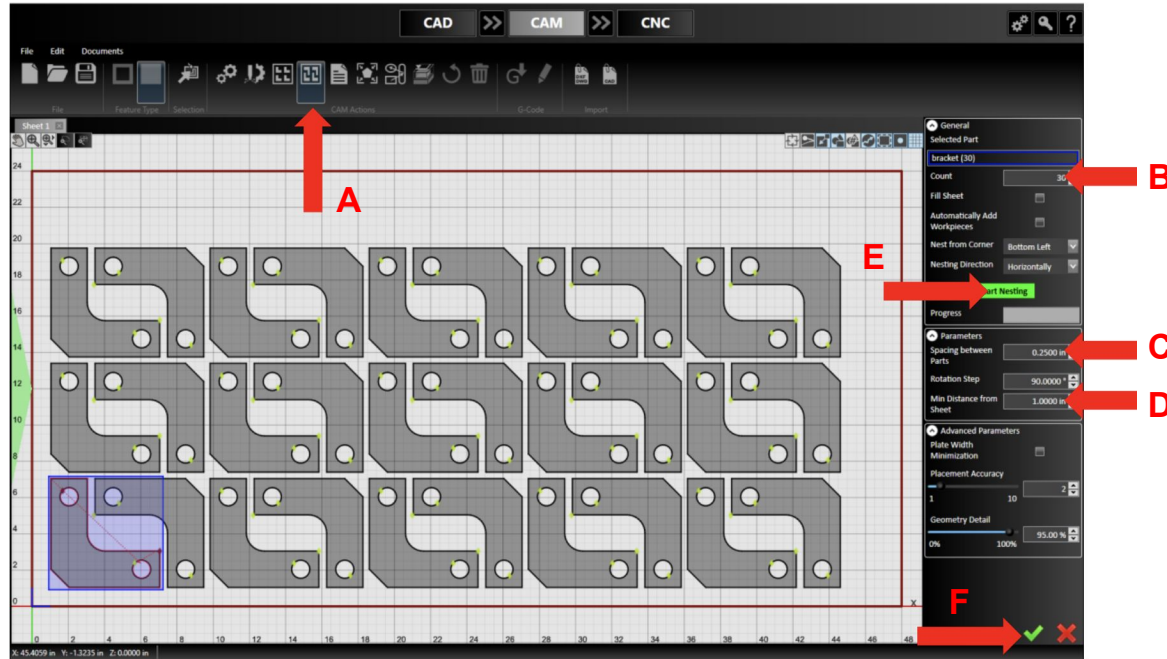
Step 5: True-Shape Nesting.

A. Select the *True-Shape Nesting* button in the *CAM Actions* section

B. Set *Count* to 30

C. In the *Parameters* section, set spacing to 0.25"

D. Set *Min Distance from Sheet* to 1"

E. Click *Start Nesting*

F. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE

Making a Toolpath in CAM

Step 6: Nesting Direction.

- A. Remain in *True-Shape Nesting*
- B. Set *Count* to 14
- C. In the *Parameters* section, set spacing to 0.25"
- D. Set *Min Distance from Sheet* to 1"

Nesting Direction: Horizontally
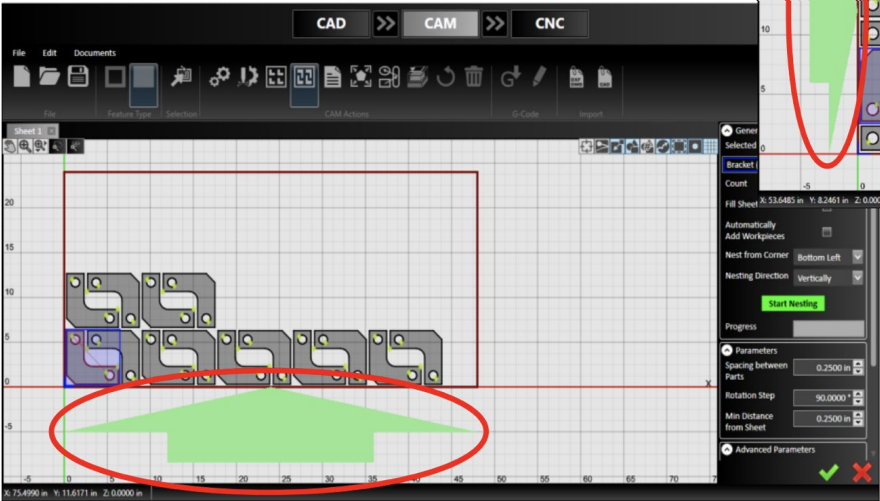
Nesting Direction: Vertically
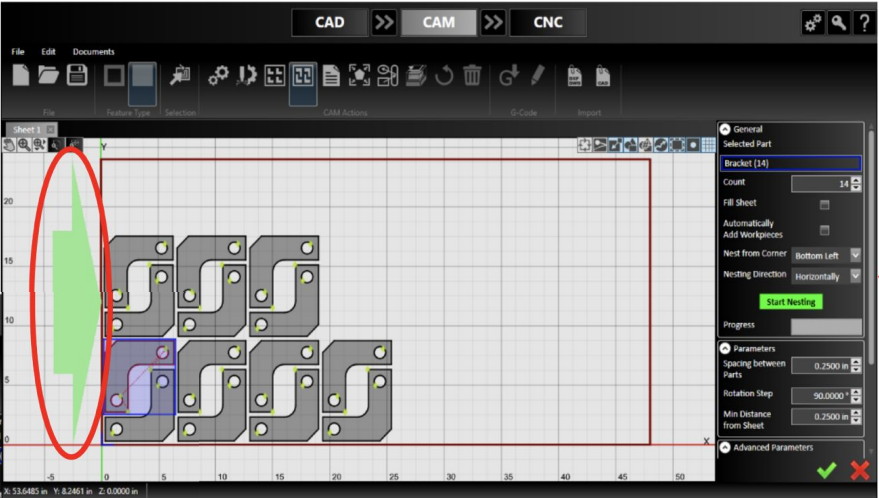
E. Locate the property *Nesting Direction* and experiment with each option
   - a. Note how the direction determines the manner in which each piece fills the sheet, starting at the corner specified in *Nest from Corner*
F. Click *Start Nesting*
G. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE

Making a Toolpath in CAM

Step 6: Transform from CAD/CAM File to CNC.

A. Click *Save* icon to save CAD/CAM file

B. Hit the double arrow button between CAM and CNC

C. G-Code will be generated for the toolpath in CAM and the CNC panel will automatically open
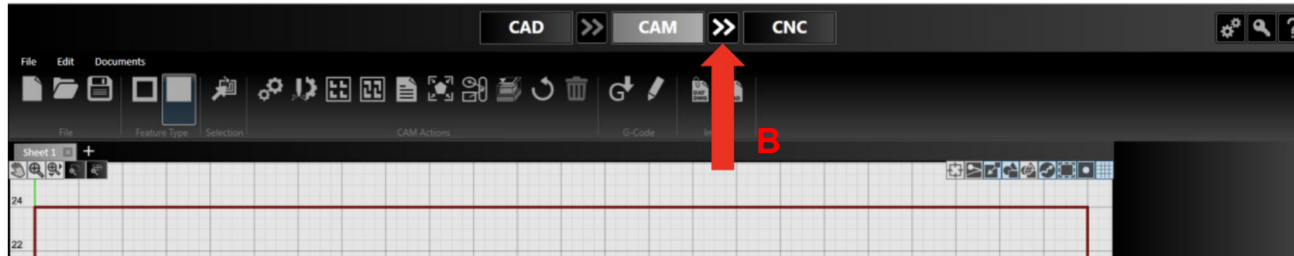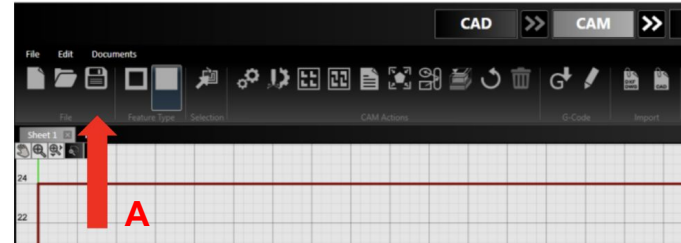
Note: If you click the CNC button, the CNC panel will open, but it will not load the G-Code for the current toolpath in CAM

FLASHCUT CNC
CONTROL MADE SIMPLE

Lesson 4:
Shape Library
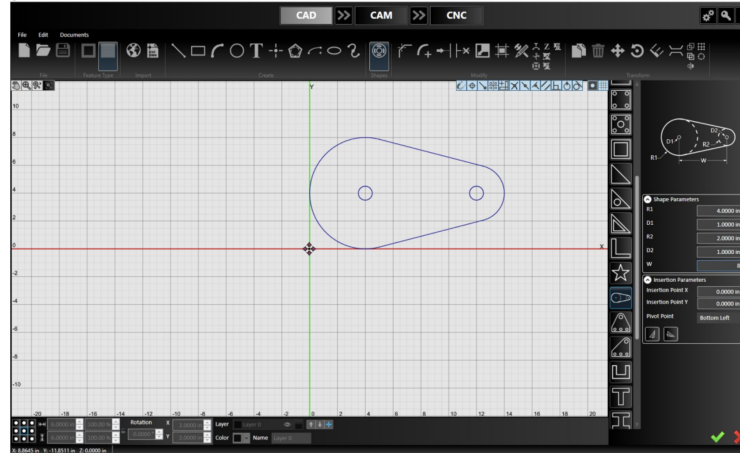
Exploring Shape Library
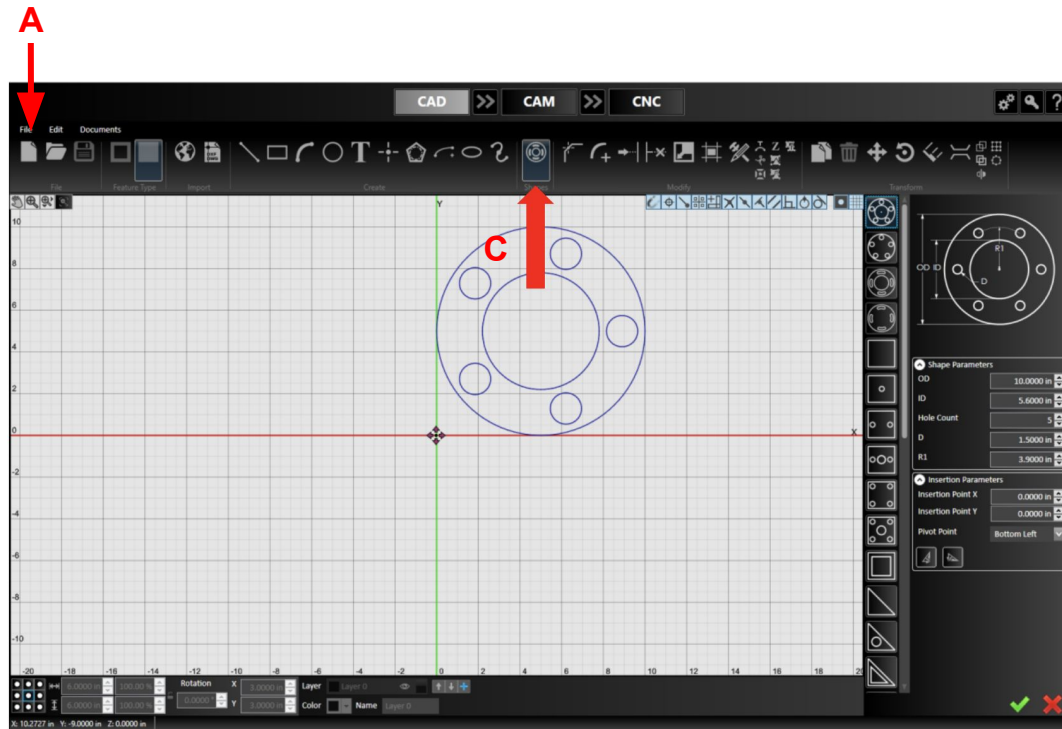
Step 1: Open a New File

A. Click the *New File* icon <u>or</u> select *New* from the *File* menu

B. Choose *2D* for the file type

C. Select the *Shape Tool*

Exploring Shape Library

Note: Once completed shape is created, they can then be edited using any of the drawing tools
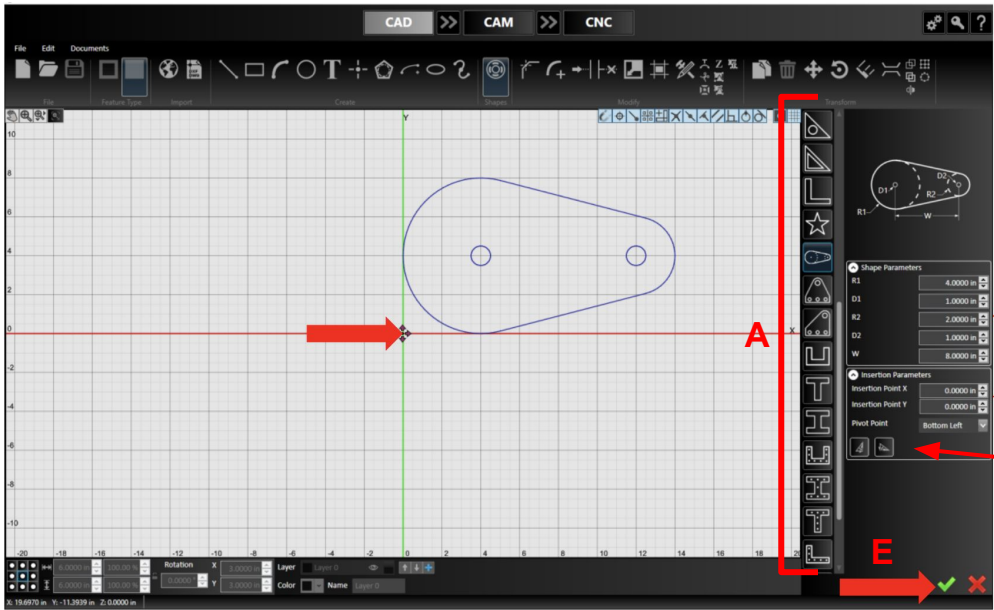
Step 2: Customize a Shape

A. Scroll through the given shapes and choose one

B. Edit the shape properties in the parameters panel

C. Choose a *Pivot Point* location

D. Drag the shape with your mouse or type the XY coordinates into the boxes labelled *Insertion Point X/Y* to specify the shape's location

E. Check it in

PRO

FLASHCUT CNC
CONTROL MADE SIMPLE

Exploring Shape Library
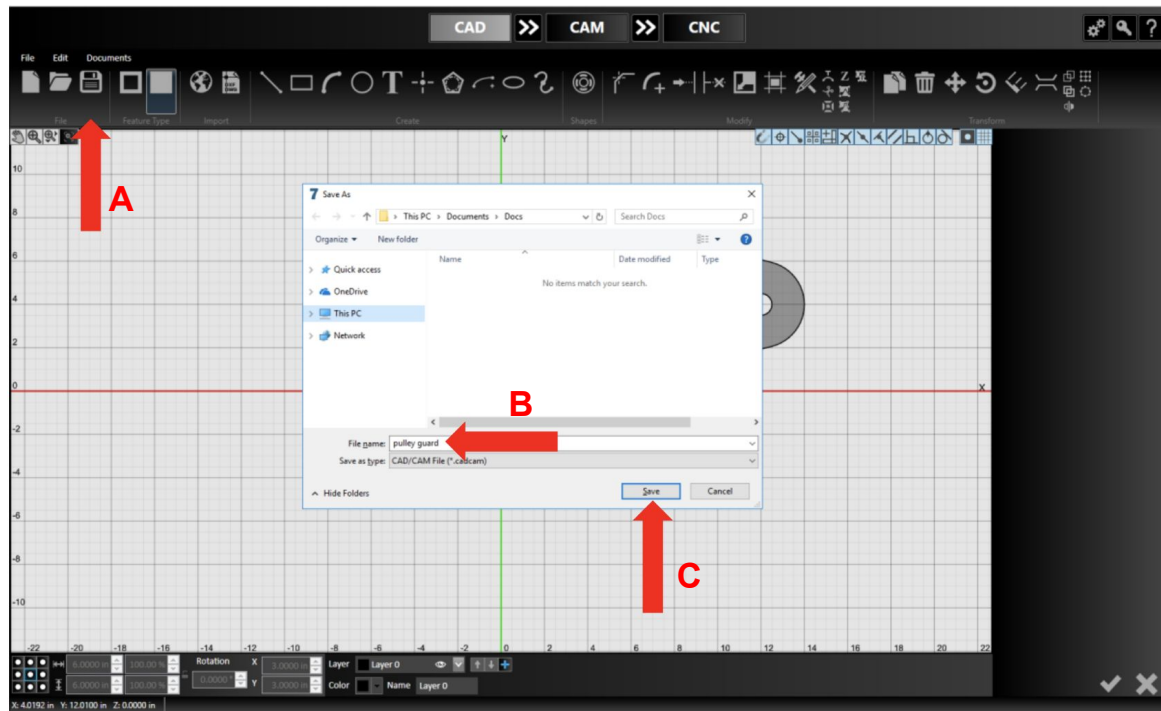
Step 3: Save the Drawing

A. Select the *Save* icon <u>or</u> from the *File* menu click *Save*

B. Name the file after your customized shape

C. Hit the *Save* button

Lesson 5:
Image Import
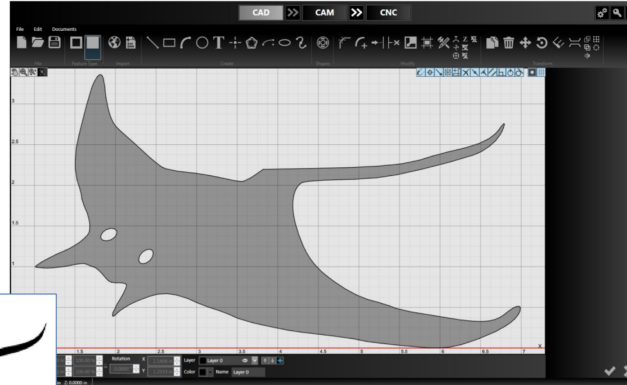
Importing Silhouette Images

Step 1: Convert to Geometry

A. Choose the *Import Silhouette* icon from the *Image Import* dropdown

B. Browse to an image file (See page 27 for acceptable file types, and example of silhouette image)

C. Alter the properties in the parameters panel to your liking

D. Check it in

FLASHCUT CNC
CONTROL MADE SIMPLE
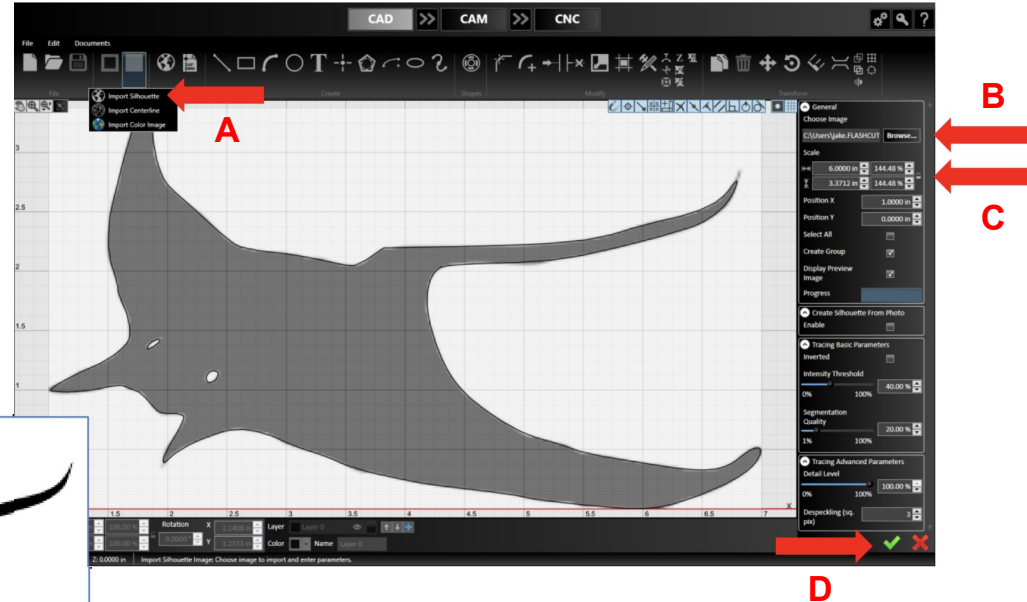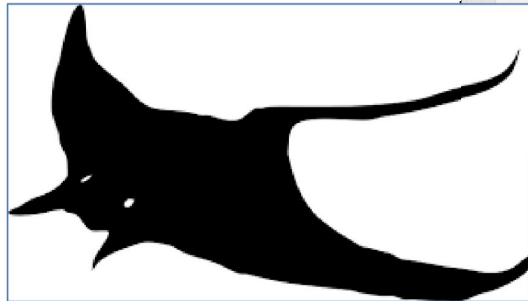
Importing Silhouette Images

Step 2: Edit the Geometry

A. Click the *Ungroup* button in the toolbar and select your image
   a. This will make the shape editable as individual line segments
   b. Use alt+click on any line segment to "chain select" an entire feature
B. Delete the eyes
C. Choose the *Ellipse Tool* and replace the eyes with ellipses

Importing Silhouette Images
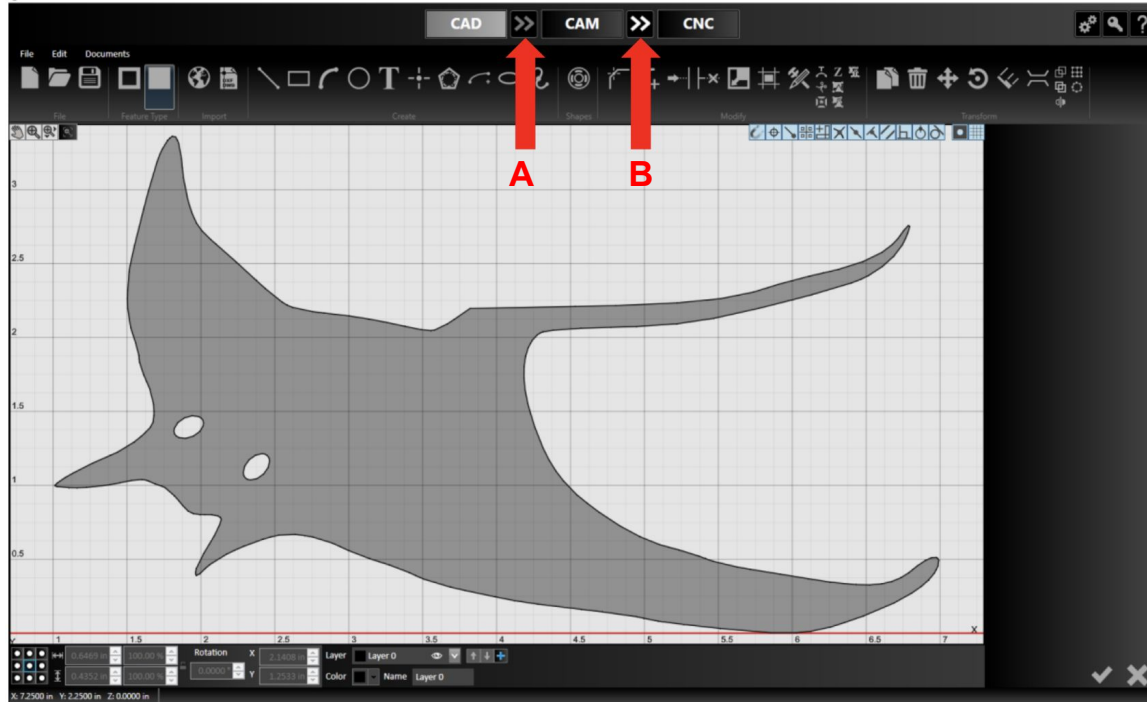
Step 3: Create the Toolpath

A. Hit the double arrows between CAD and CAM to push the drawing to CAM and automatically create a tool path

B. Hit the double arrows icon to generate G-Code and send it to the CNC module

FLASHCUT CNC
CONTROL MADE SIMPLE

Importing Silhouette Images
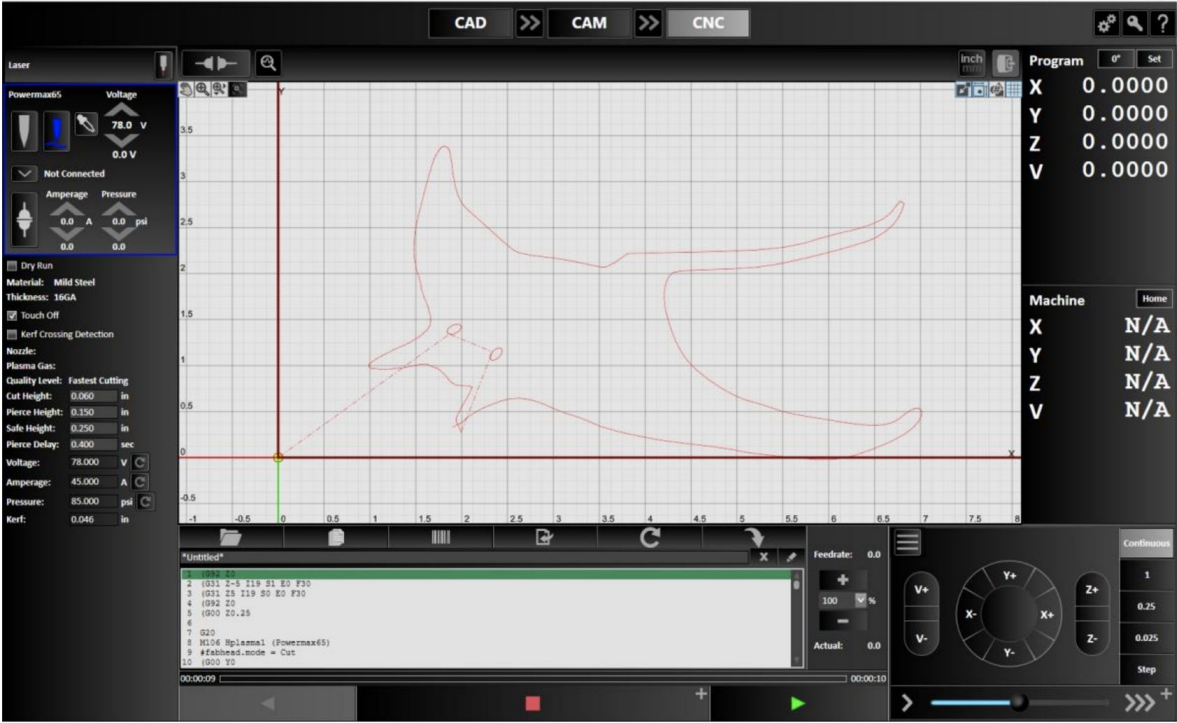
Step 4: Cut the Parts in CNC

Importing Color Images
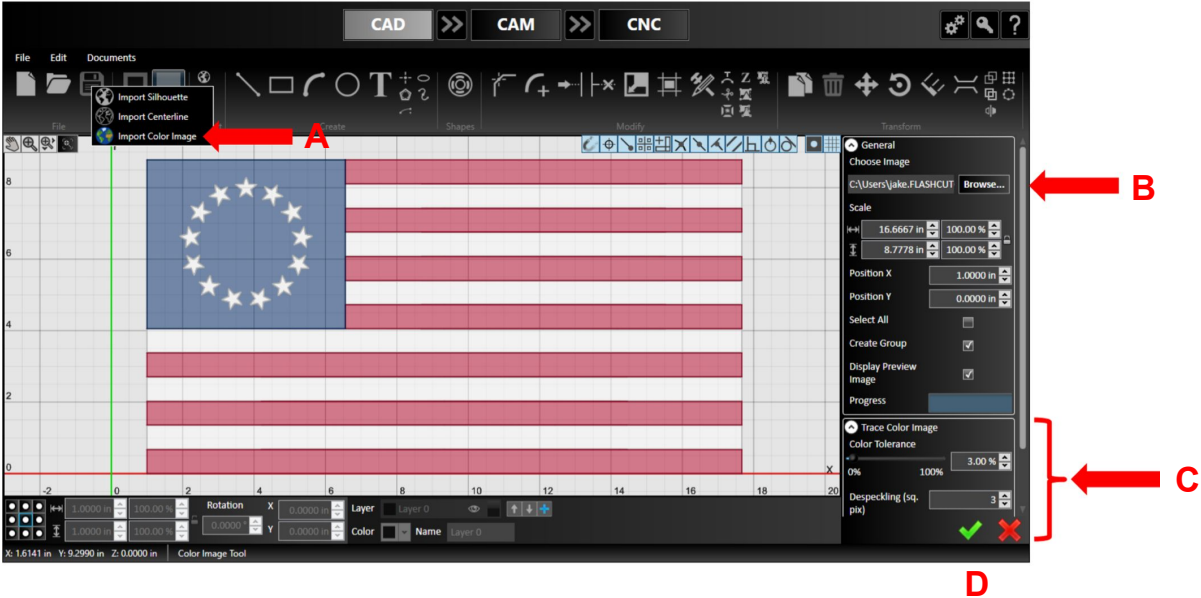
Step 1: Convert to Geometry

A. Choose the *Import Color Image* icon from the *Image Import* dropdown

B. Browse to an image file (See page 27 for acceptable file types, and example of silhouette image)

   a. Images with strong contrast work best

C. Alter the properties in the parameters panel to your liking

D. Check it in

Importing Color Images
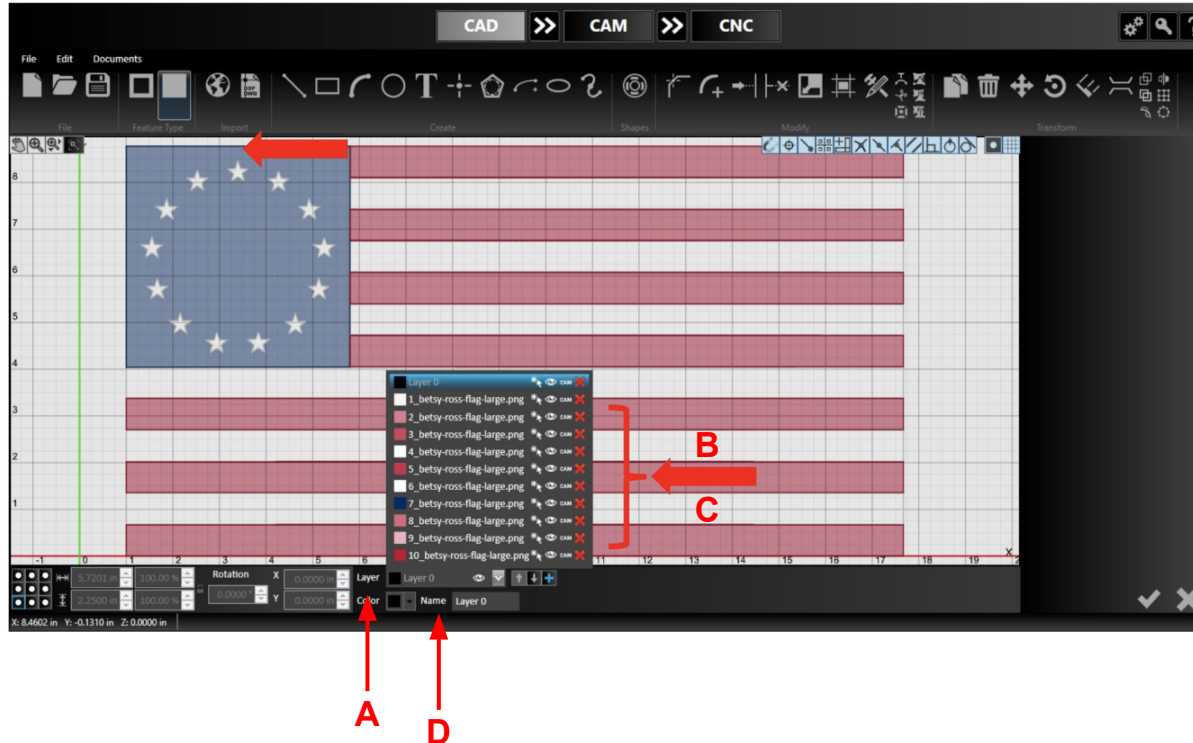
Step 2: Organize Layers

A. Click on the *Layer Tool* at the bar on the bottom of the screen

B. Click on the visibility icon of each layer to determine what geometry is associate with each layer

C. Delete the layers that are not necessary

D. Click on an individual layer and rename it by typing in the name field

Importing Color Images
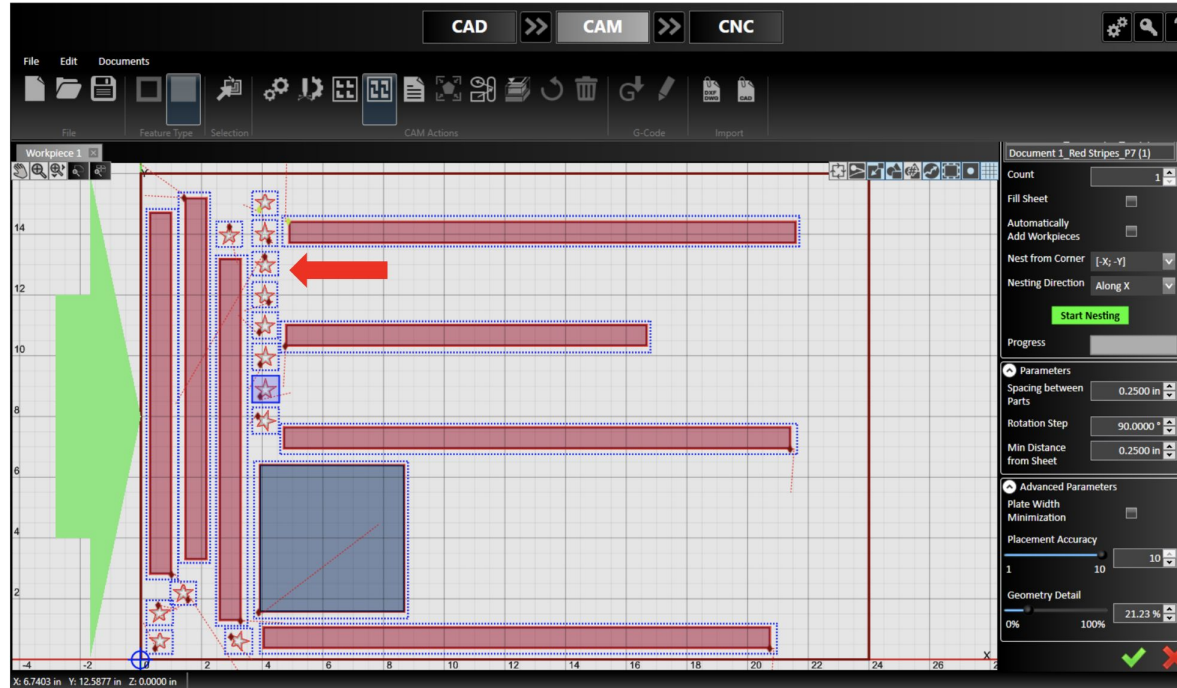
Step 3: Nest the Parts

A. Go to CAM

B. Follow the steps previously executed for *Nesting*

    a. Notice how the stars fill the sheet as individual parts, which may not be desirable

Importing Color Images

Step 4: Combine Layers

A. Return to CAD
B. Choose the white star layer; the stars should be highlighted
C. In the layer tool, choose the blue star field layer; this will place the highlighted geometry (in this case, the stars) in the blue star field layer
   a. The stars are not cutouts of the blue layer
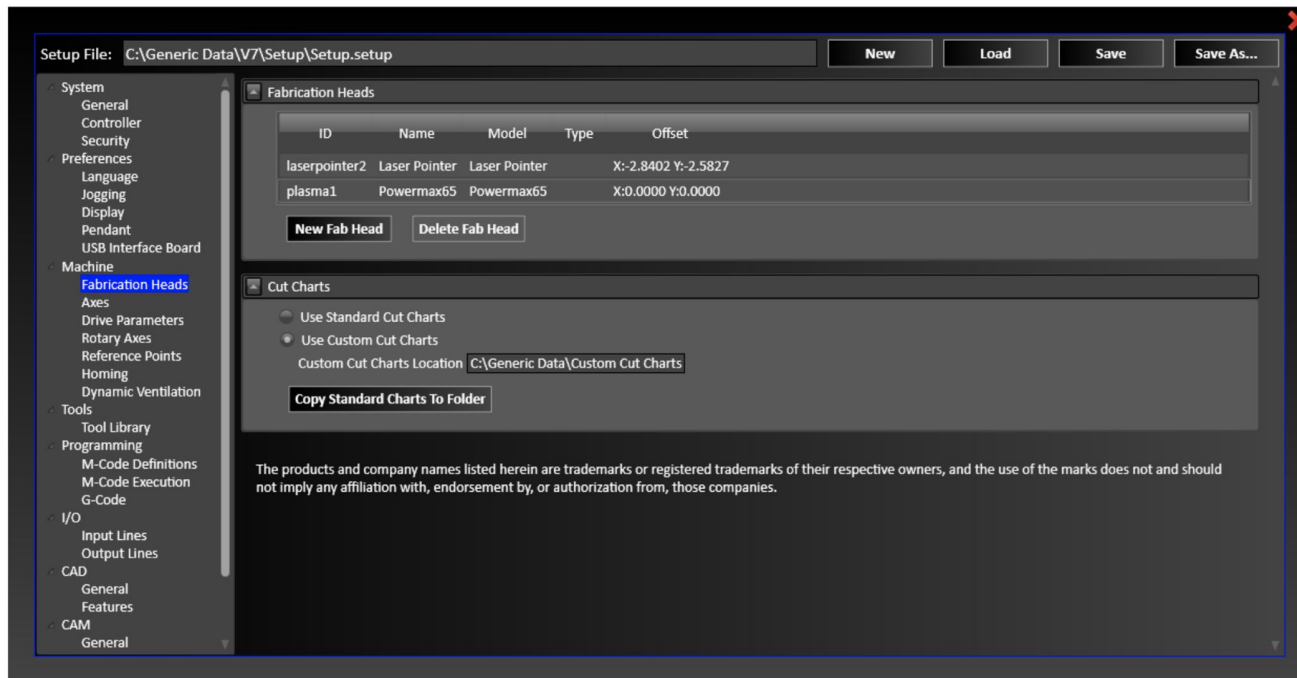
Importing Color Images
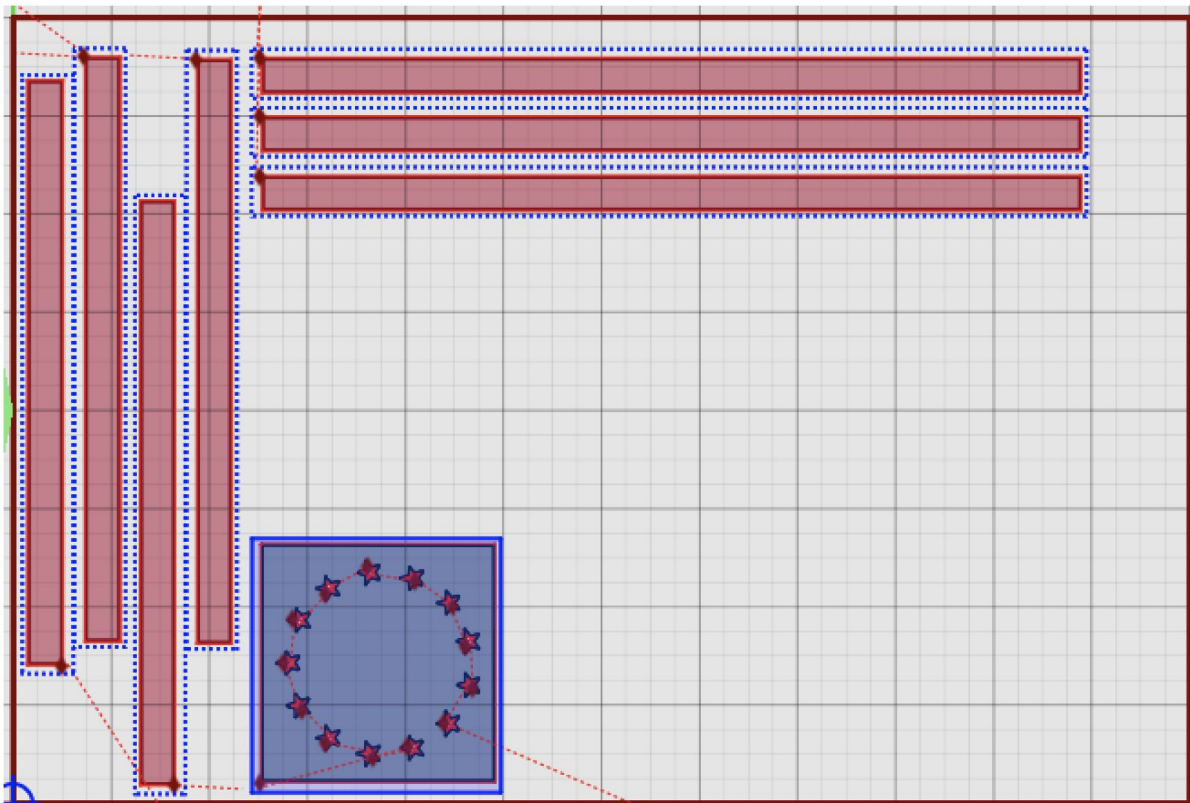
Step 5: Nest the Parts Again

A. Go to CAM
B. Follow the steps previously executed for *Nesting*
   a. Notice how the stars are no longer individual parts, but cutouts in the star field layer

Lesson 6:
Fabrication Heads & Cut Charts

Adding a Fab Head

- Fabrication Heads are the actual cutting heads on your machine
- Plasma, Water Jet, Oxy Fuel, Router, Marker, Laser, and/or Laser Pointer
- Some types of fab heads 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
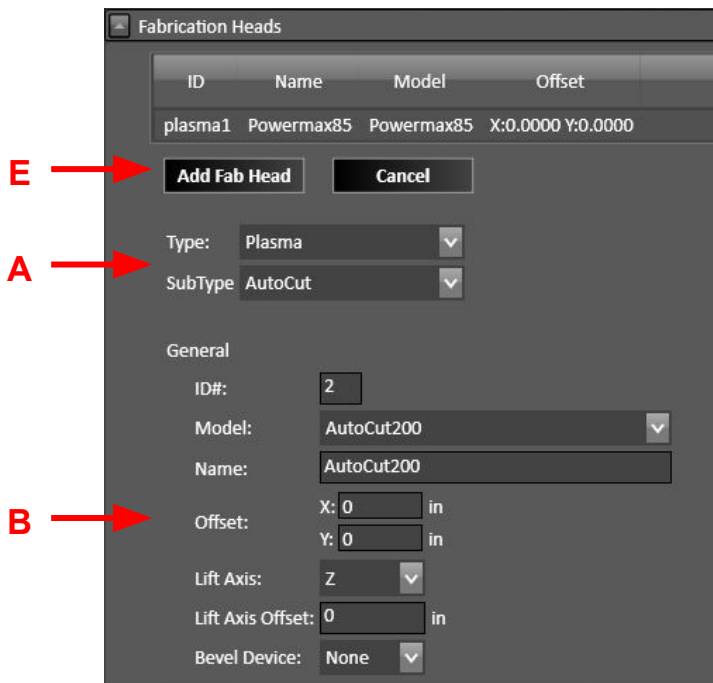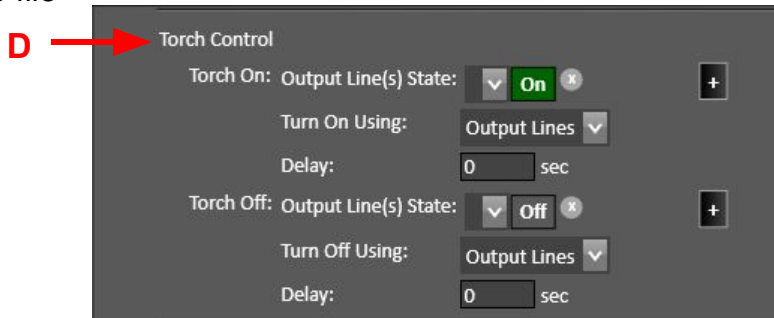
Adding a Fab Head
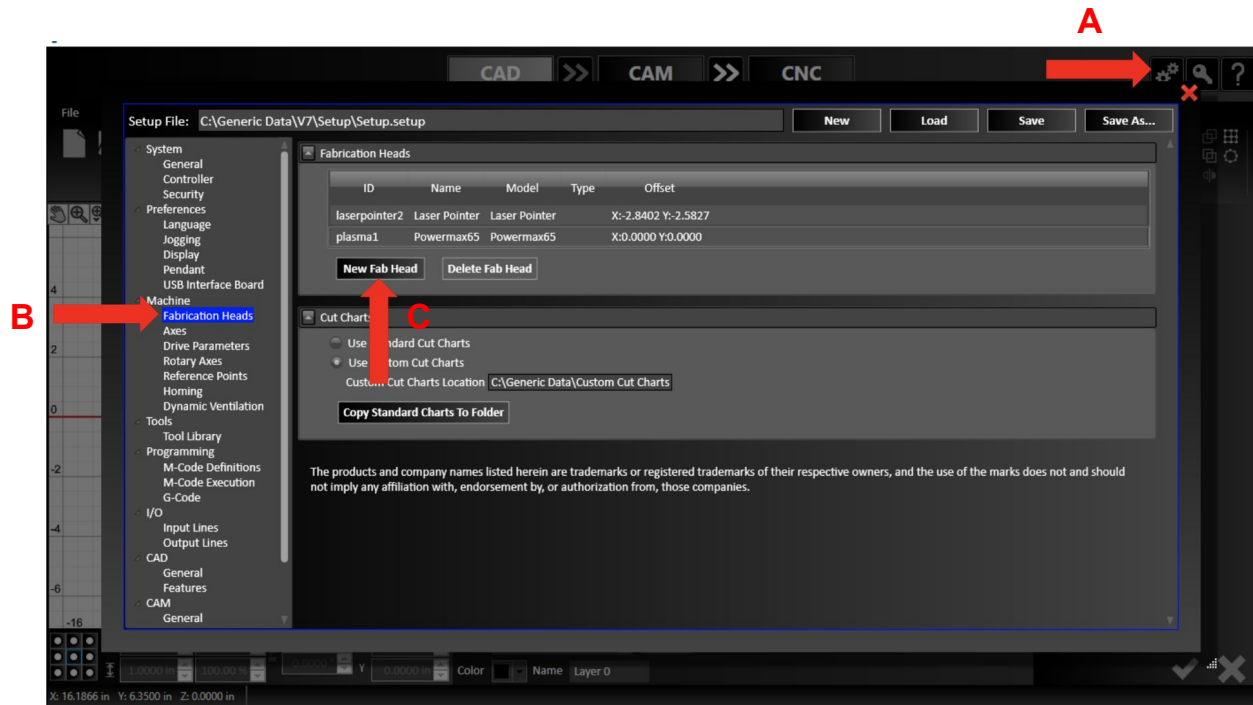
A. Click the *Gears* icon located at the top right corner of the window
   a. The Configuration Screen will pop up
B. Go to the *Fabrication Heads* section
C. Select a *New Fab Head*

Adding a Plasma Fab Head

A. Choose the *Type, SubType*, and *Model* from the dropdown menus; *ID* and *Name* will be automatically populated
B. Set X/Y *Offset* if you have more than one fab head
C. Select RS-232/485 and a COM Port if you have direct communication to set the torch's current and pressure
D. Fill in the proper *Output Lines* for turning the *Torch On* and *Off* as well as the correct macros for Automatic Execution Mode
E. Hit the *Add Fab Head* button to finalize your fab head changes
F. *Save* the Setup file

E ➡

A ➡

B ➡

D ➡

Fabrication Heads

ID	Name	Model	Offset
plasma1	Powermax85	Powermax85	X:0.0000 Y:0.0000

Add Fab Head   **Cancel**

Type: Plasma

SubType AutoCut

General

ID#: 2

Model: AutoCut200

Name: AutoCut200

Offset: X: 0 in
         Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Bevel Device: None

Torch Control

Torch On: Output Line(s) State: On ⊗ +

Turn On Using: Output Lines

Delay: 0 sec

Torch Off: Output Line(s) State: Off ⊗ +

Turn Off Using: Output Lines

Delay: 0 sec

FLASHCUT CNC
CONTROL MADE SIMPLE

Adding a Plasma Fab Head (THC)

If you have *Torch Height Control*, check the box and fill in the parameters:

A. *Arc Voltage Divisor* – This is typically 50 and must match the exact voltage divider setting in your torch. For THC to work properly: (**Maximum Raw Voltage ÷ Arc Voltage Divisor**) must be < 6V.

B. *Lock Out Factor* – THC will suspend to prevent unwanted diving when the XY velocity of the torch is below this percentage of the feedrate

C. *Lock Out Hysteresis* – THC will resume when the XY velocity goes above: (***Lock Out Factor* + Lock Out Hysteresis**) *** Programmed Feedrate**

D. *Set Point Offset* – Offsets the small idle voltage when torch is not in use

E. *Sensitivity* & *Deadband Voltage* – This is a factor that determines how quickly the THC acts towards voltage fluctuation. This number can vary from 0.09V-1.94V, smaller values being more sensitive

Adding a Plasma Fab Head (Sensing)

Sensing will allow the torch to find the surface of the plate

A.   Choose the sensing *Direction* and *Feedrate*

B.   Choose the *Monitor Input Line* for the *Primary Sensor*, typically an Ohmic sensor

C.   *Sensor Offset* is the extra distance that the Z axis travels between zero and tripping the sensor

D.   *Use Additional Sensor* enables a backup switch in case the primary touch off method fails; this is typically a mechanical switch or proximity sensor

E.   *Program Zero Sensing* enables the torch to set program zero at the sheet

F.   *Retract Distance* is amount that the torch will raise after sensing

Adding a Plasma Fab Head (Cont.)

A. *Safe Height* will raise the torch to a specified height after every cut
    a. This height can be in Program or Machine Coordinates
B. *Touch Off* causes the torch to sense the plate before every cut
C. If *Zero Life Axis Program Coord* in enabled, each tough off will also zero the Z axis program coordinate
D. *Rapid Move to Start Height* will cause the Z axis to rapid down to a specified height before beginning the sensing procedure while touching off
E. If *Enable Suppression Radius* is checked, the torch will skip all touch off procedures within a specified radius of the previous touch off location
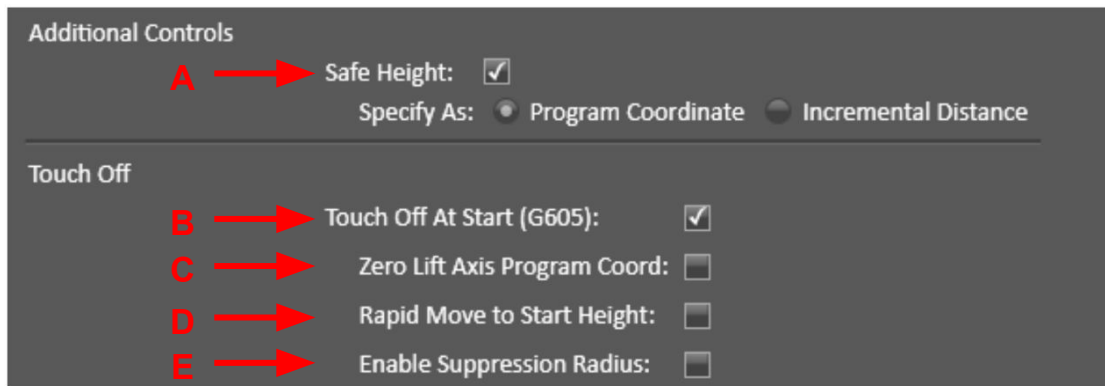
The radius is specified in the CNC panel at run time

Advanced Plasma Control

Cut leadin when returning to toolpath (found under *Cutting* when customizing a new fab head) – Designed for recovering from a lost torch using the following steps:

A. Use jump to line and the run forward, backward, and feedhold controls in *Dry Run* mode to the point the cut stopped
    a. *Dry Run* mode can be toggled in CNC on the left panel
B. Jog off of the toolpath to a point that will be off the finished part
C. Disable *Dry Run* mode and start the program from that point
D. 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
E. The program will proceed normally from that point
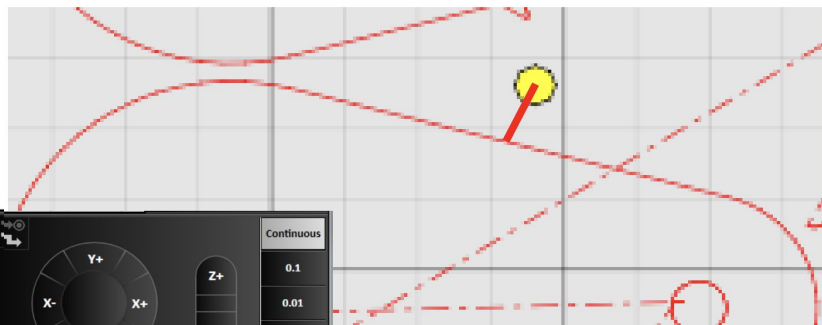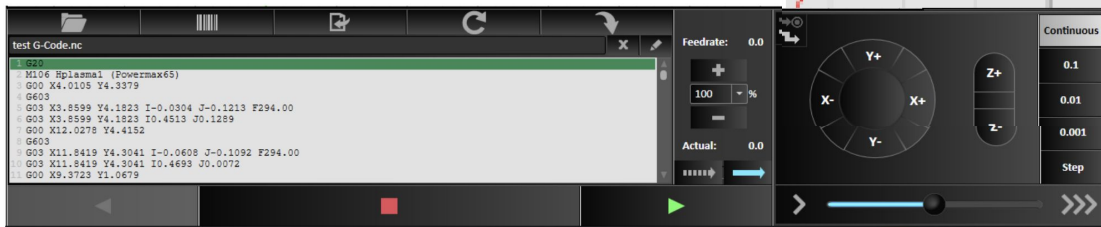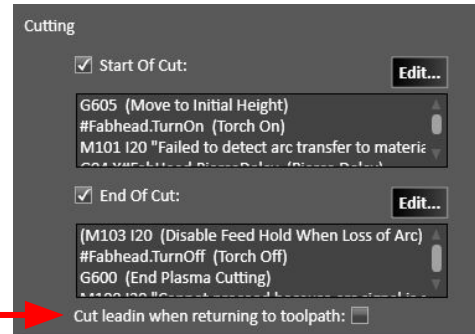
Cut Charts

A. Many of the Plasma and Oxy Fuel fab heads have pre-loaded cut charts with manufacturer-recommended cut parameters
B. You can use the Standard Cut Charts for the fab head manufacturer's recommended settings
C. If you choose the Custom Cut Chart option, then the standard cut charts will be stored to the specified location as several CSV file copies when you hit the *Copy Standard Charts To Folder* button
D. Any value in these cut charts can be edited using a spreadsheet program such as Excel
E. Feedrates and pierce times for waterjet are calculated from a formula embedded in the software

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	NozzleModel	MaterialType	Thickness	KerfWidth	CutHeight	Pierce Height	Pierce Delay	BestFeedrate	BestVoltage	Production Feedrate	Production Voltage	Current	Pressure	EdgeStart	Units
2	85A Shielded	Mild Steel	0.1345	0.068	0.06	0.15	0	250	122	336	121	85	85	N	English
3	85A Shielded	Mild Steel	0.1875	0.071	0.06	0.15	0.2	185	123	220	123	85	85	N	English
4	85A Shielded	Mild Steel	0.25	0.073	0.06	0.15	0.5	130	123	160	126	85	85	N	English
5	85A Shielded	Mild Steel	0.375	0.078	0.06	0.15	0.5	85	126	86	127	85	85	N	English
6	85A Shielded	Mild Steel	0.5	0.09	0.06	0.18									
7	85A Shielded	Mild Steel	0.625	0.095	0.06	0.18									
8	85A Shielded	Mild Steel	0.75	0.1	0.06	0.24									
9	85A Shielded	Mild Steel	0.875	0.105	0.06	0.06									
10	85A Shielded	Mild Steel	1	0.11	0.06	0.06									
11	85A Shielded	Mild Steel	1.125	0.115	0.06	0.06									
12	85A Shielded	Mild Steel	1.25	0.12	0.06	0.06									
13	85A Shielded	Stainless Steel	0.1406	0.068	0.06	0.15									
14	85A Shielded	Stainless Steel	0.1875	0.071	0.06	0.15	0.2	200	122	240	121	85	85 N		English

Cut Charts
- Use Standard Cut Charts
- Use Custom Cut Charts

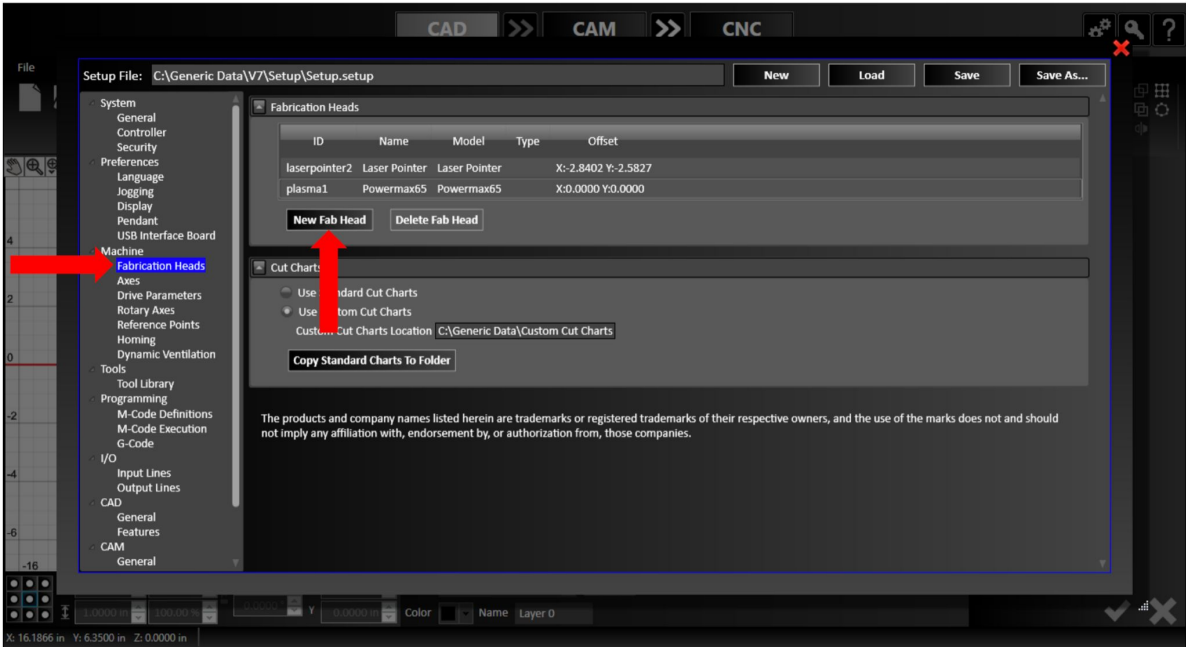Custom Cut Charts Location C:\Generic Data\Custom Cut Charts

Copy Standard Charts To Folder

FLASHCUT CNC
CONTROL MADE SIMPLE

Adding a Laser Pointer

A. Go to *Fabrication Heads* in the Configuration Screen and select *New Fab Head*

Adding a Laser Pointer

A. Choose the Type *Laser Pointer*
B. *ID* and *Name* will be automatically populated
C. Set X/Y *Offset* as the distance from the center of the laser pointer to the center of the main fab head
D. Fill in the proper *Output Lines* for turning the *Torch On* and *Off* as well as the correct macros for Automatic Execution Mode
E. Hit the *Add Fab Head* button to finalize your fab head changes
F. *Save* the Setup file

Lesson 7:
Advanced CADCAM Tools
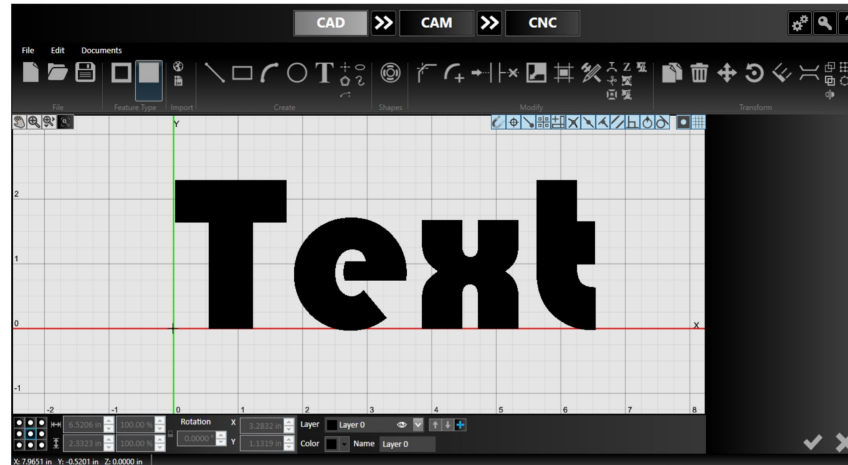
Using the *Text* Tool

A. Choose starting point with your mouse or type the XY Positions
B. Type the text you want
C. Choose a font; once you choose an initial font, you can use the up/down arrows to scroll through and view other fonts
D. Adjust the size, kerning, line spacing, angle, bold, and italic options
E. Check it in
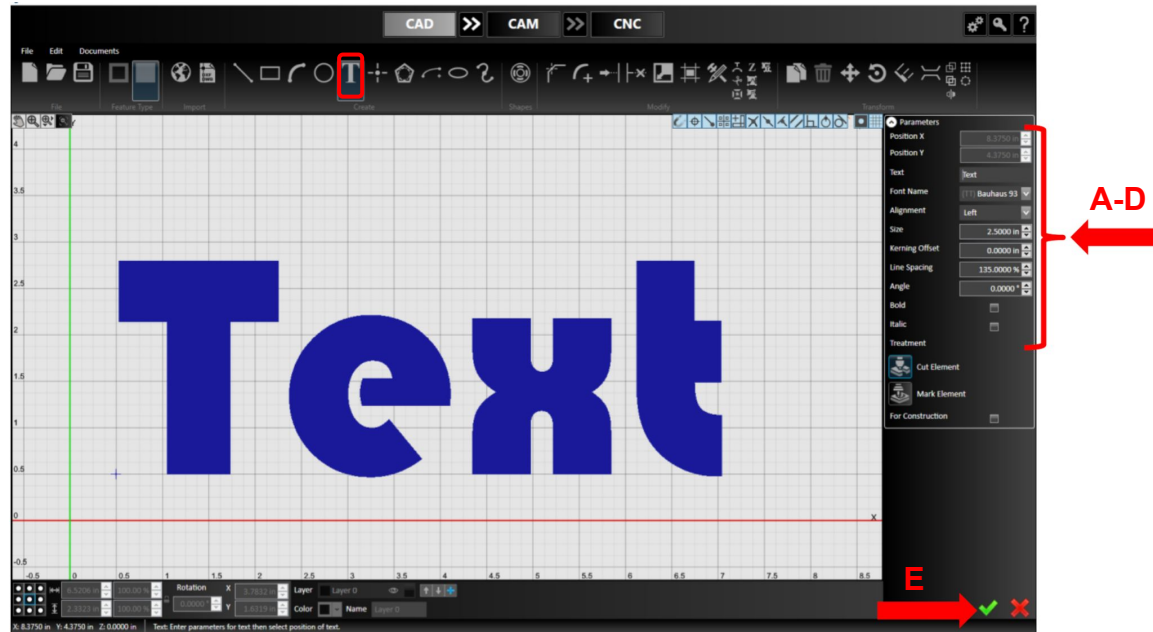
Using Advanced Text Toolpath

A. Go to the CAM mode by selecting the double arrows

B. 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)

C. You can move and change the lead-in of each letter, which will be retained even if another letter in the word is edited
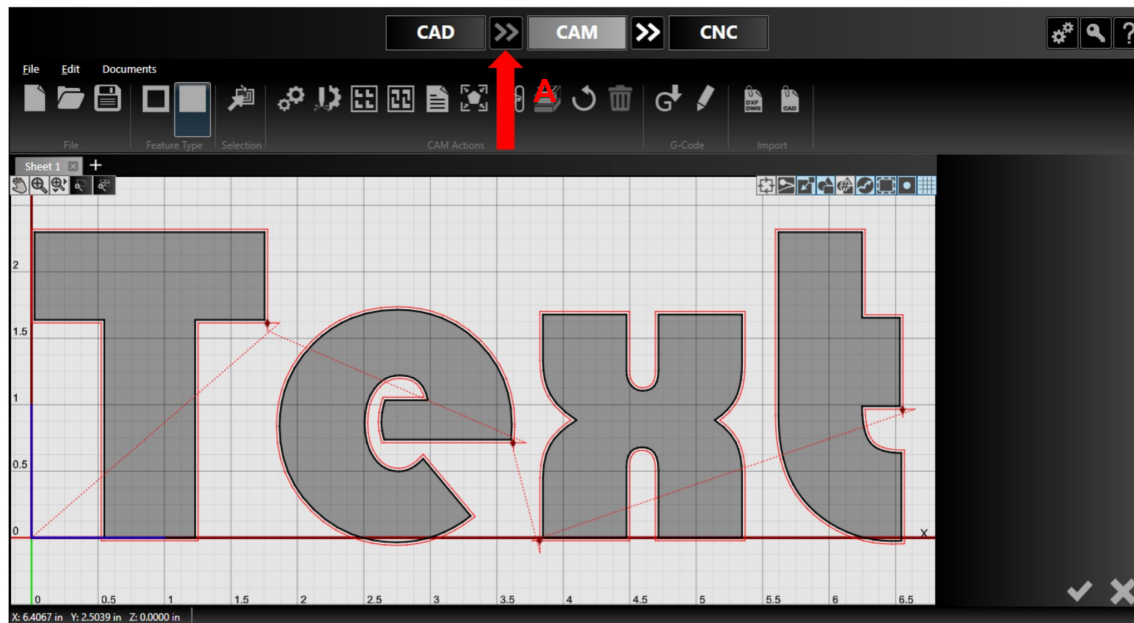
Editing Text Without Explode
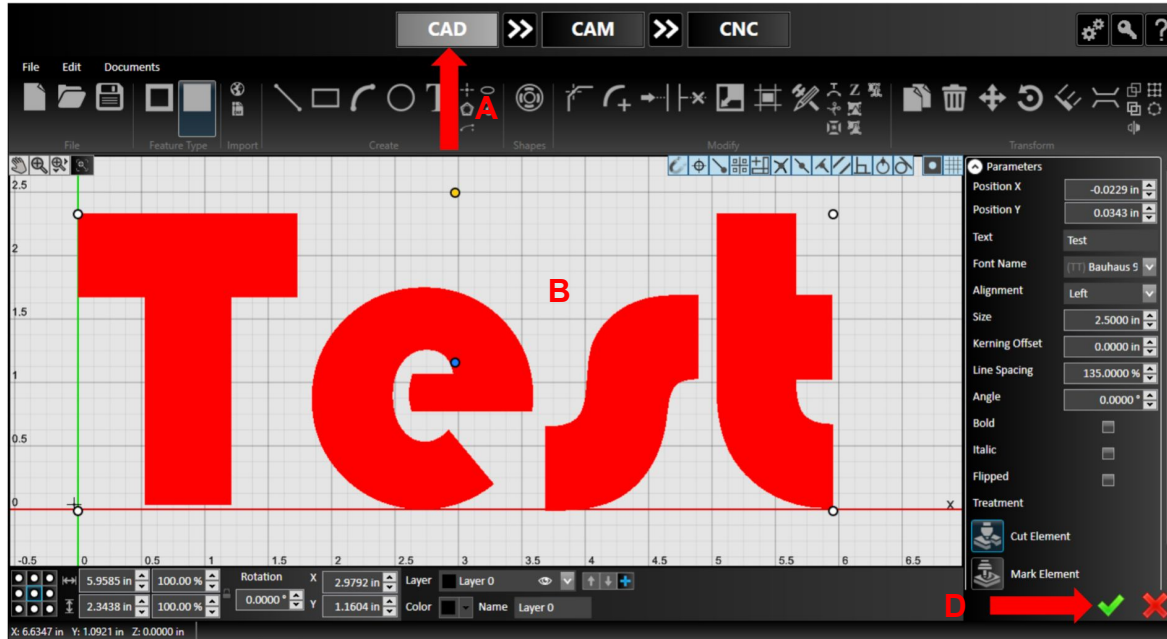
A. Return to the CAD module
B. Click on the text to open the object's parameters panel for editing, even after generating a tool path
C. Change one letter in the word you typed
D. Check in any changes

Toolpath of New Text

PRO

A. Go to the CAM module
   a. Notice how the toolpath regenerated with the new text from the font without having to explode the font

FLASHCUT CNC
CONTROL MADE SIMPLE
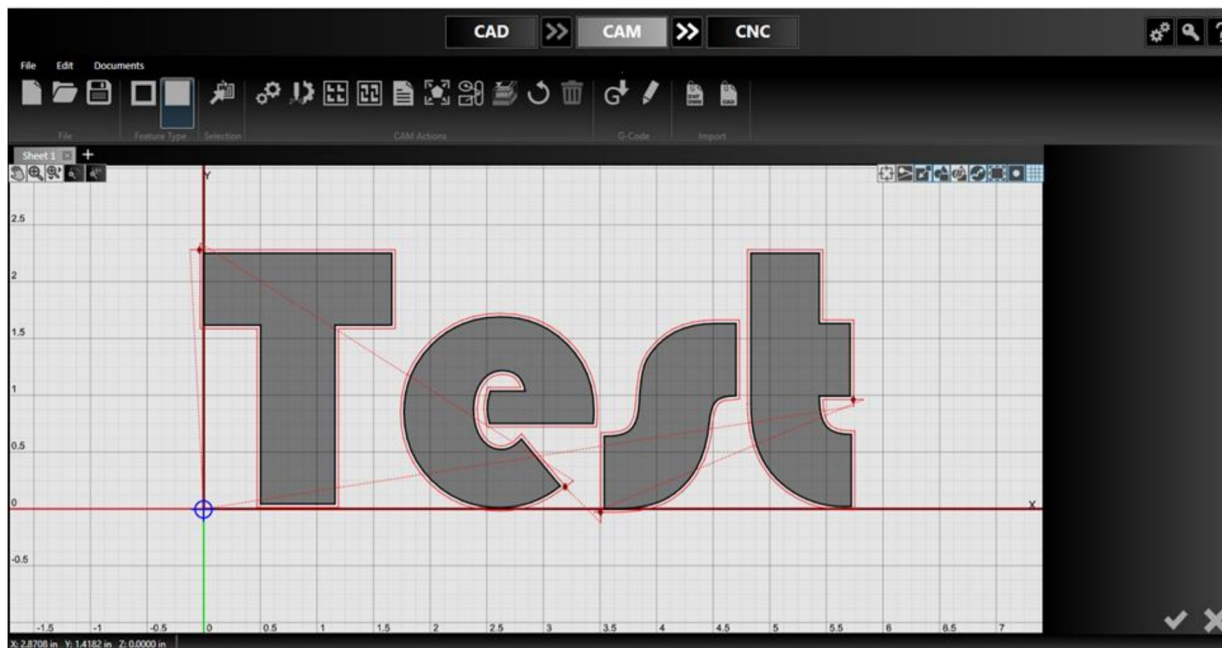
Using the *Fix* Tool
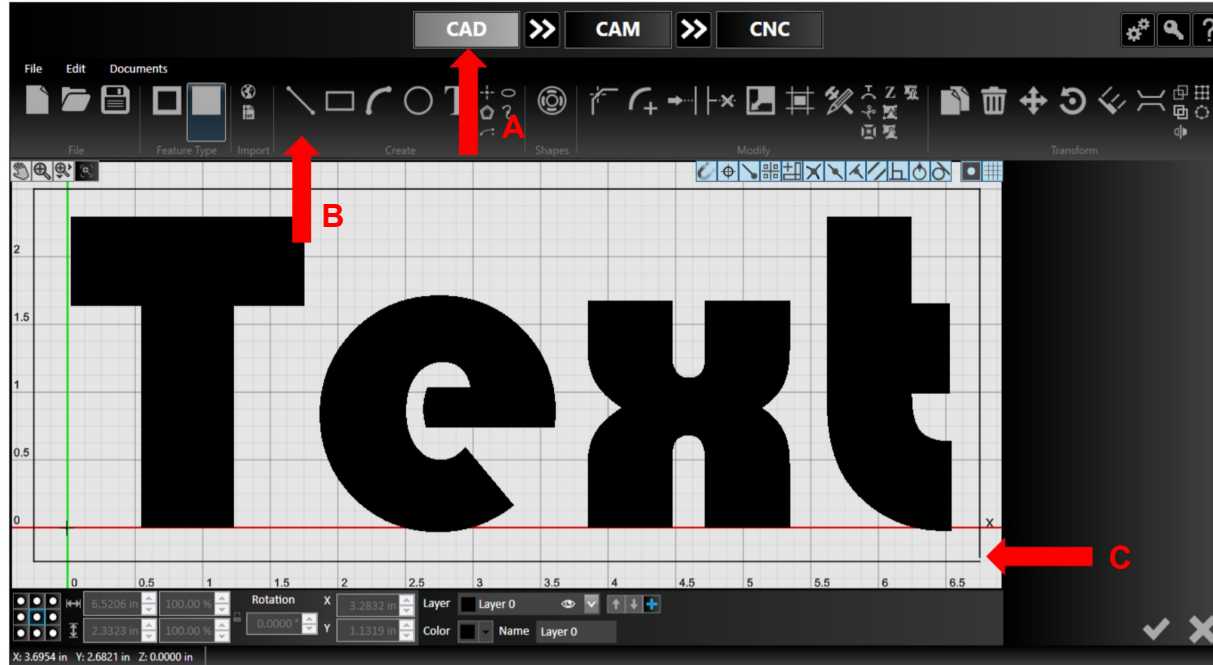
A. Go back to CAD
B. Click *Line Tool*
C. Draw box around the text, but leave the last line disconnected from the first; this is easier to do if you deactivate all of the snapping tools

Create the Toolpath

A. Go to the CAM module by hitting the double arrows
B. The toolpath for the rectangle did not appear because it is not a closed entity
C. If you choose the *Make Top Level Features a Cutout* button, the rectangle will appear but the text will become a cutout and not a part
D. Switch back to *Make the Top Level a Part* feature type
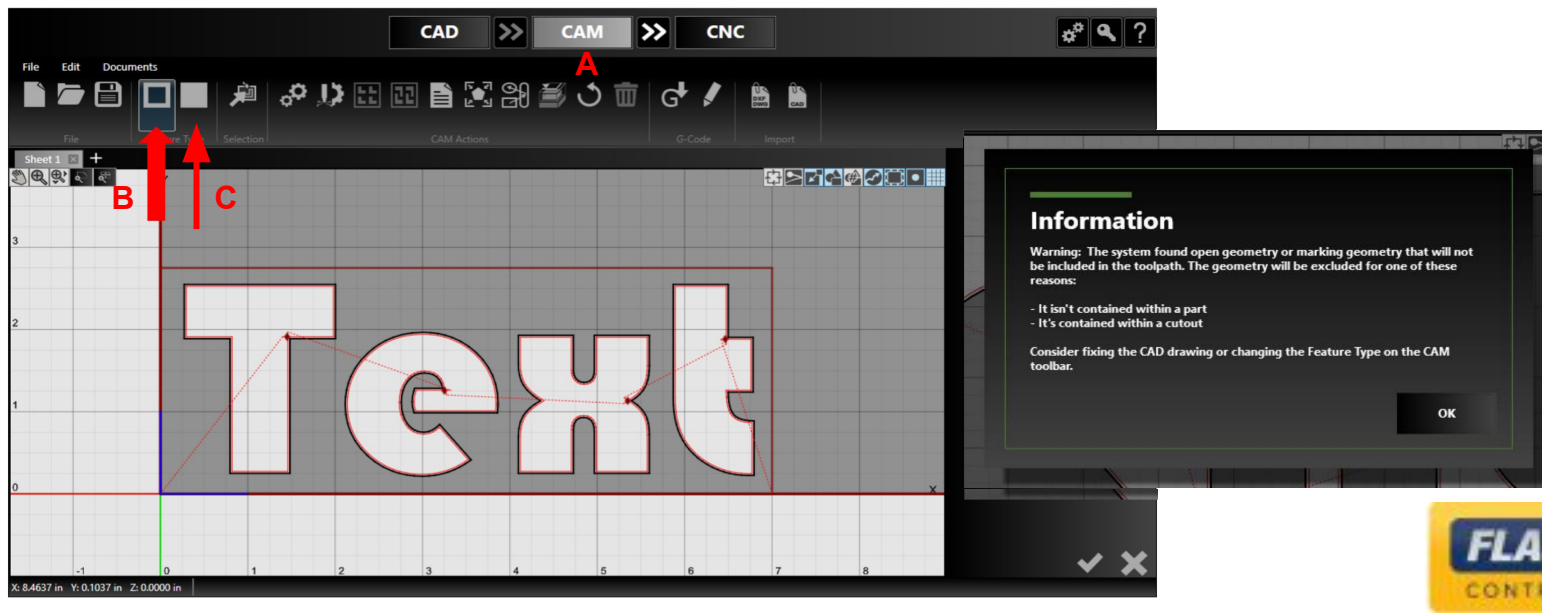
"Fix" the Rectangle
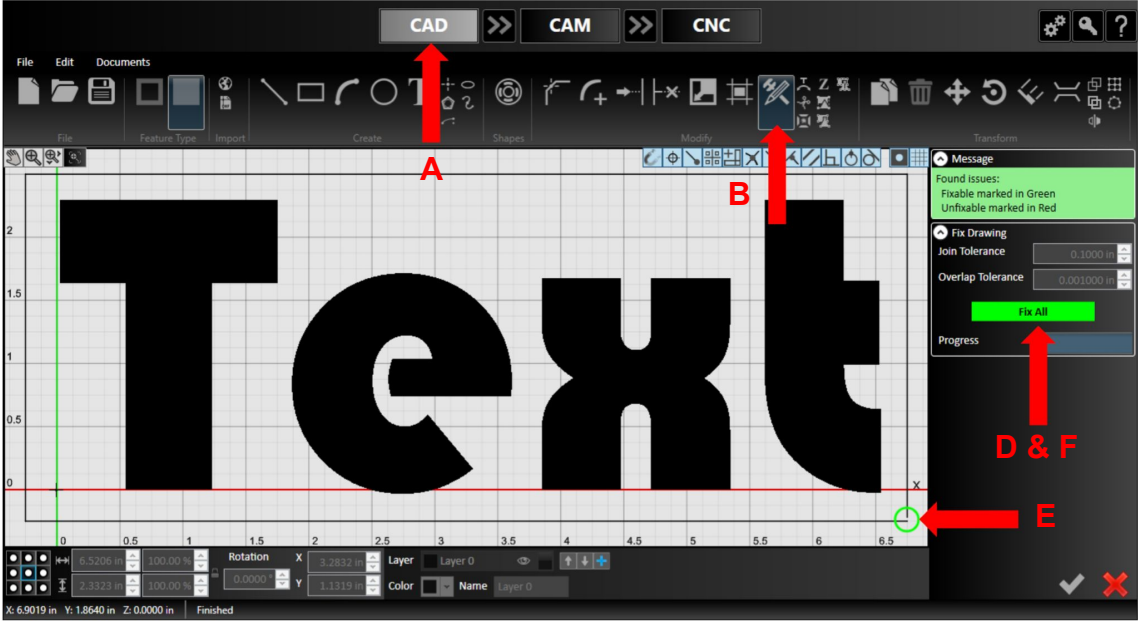
A. Go back to the CAD tab
B. Click the *Fix Drawing* tool
C. Choose a tolerance larger than any of your drawing errors
D. Hit the *Search Issues* button
E. Any issues found will be noted with a green circle; any issues that cannot be fixed automatically will be noted with a red circle
F. You can either choose the *Fix All* button to automatically fix the issue of fix them manually

Create the Toolpath
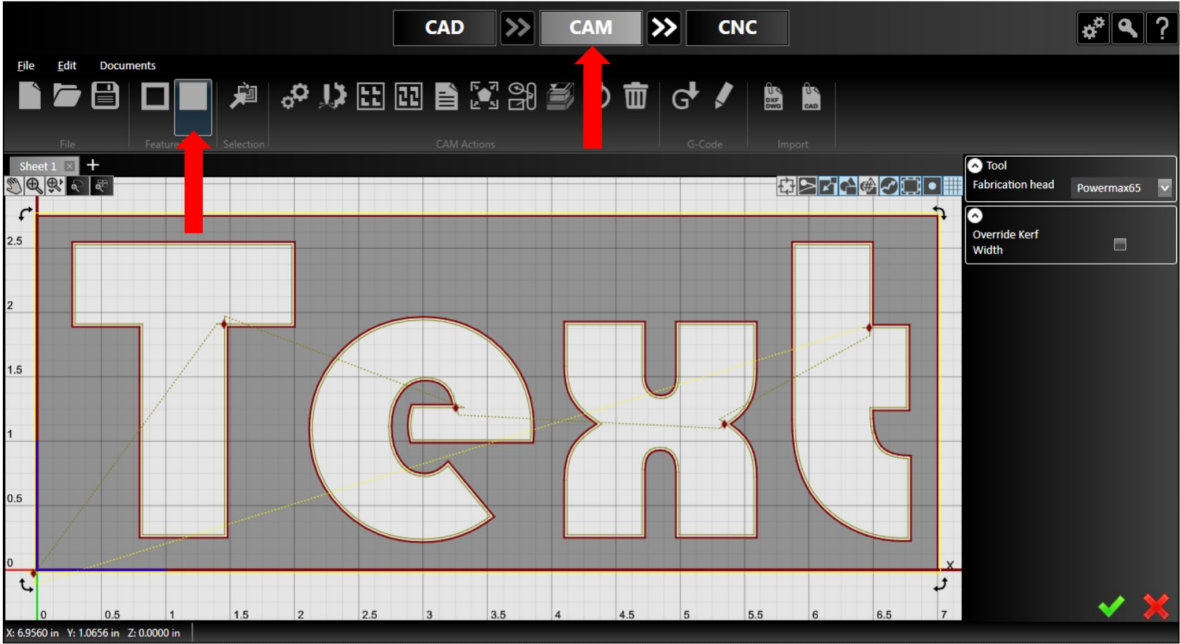
A. Go to CAM by clicking on the double arrows
   a. Note that the "fixed" rectangle now appears no matter which feature type is the top level

Using the *Measure* Tool
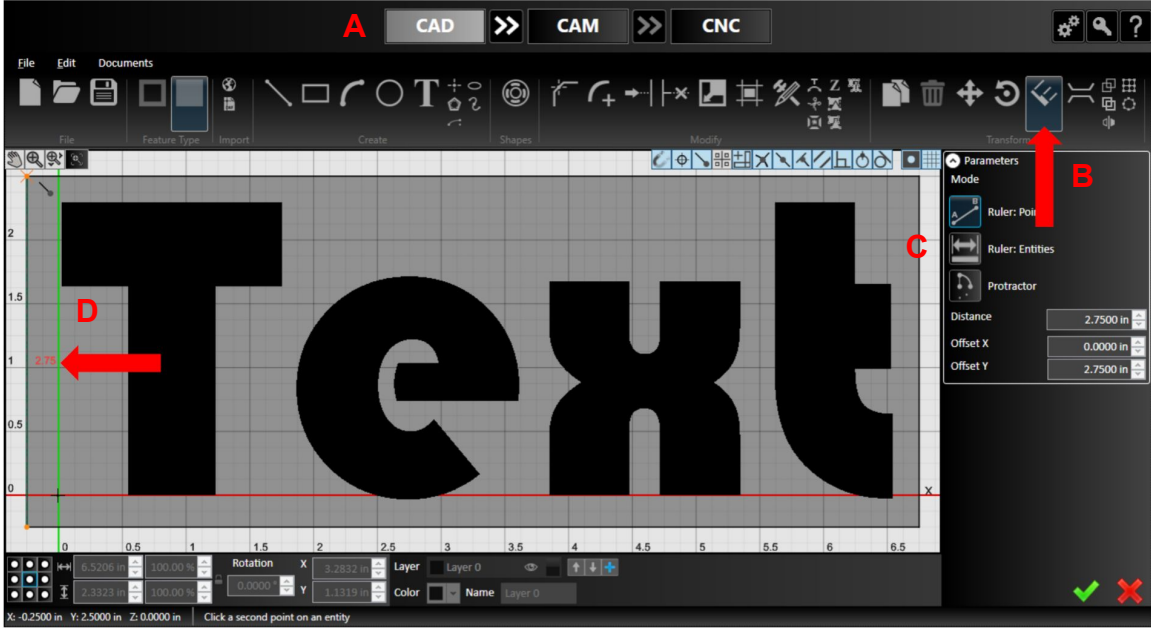
A. Go to CAD
B. Click *Measure Entities with Ruler* tool
C. Select the mode you want: *Points, Entities,* or *Protractor*
D. Select the portion of the drawing you want to measure
E. The corresponding dimensions will appear in the right side as well as on the drawing

Using Bridges
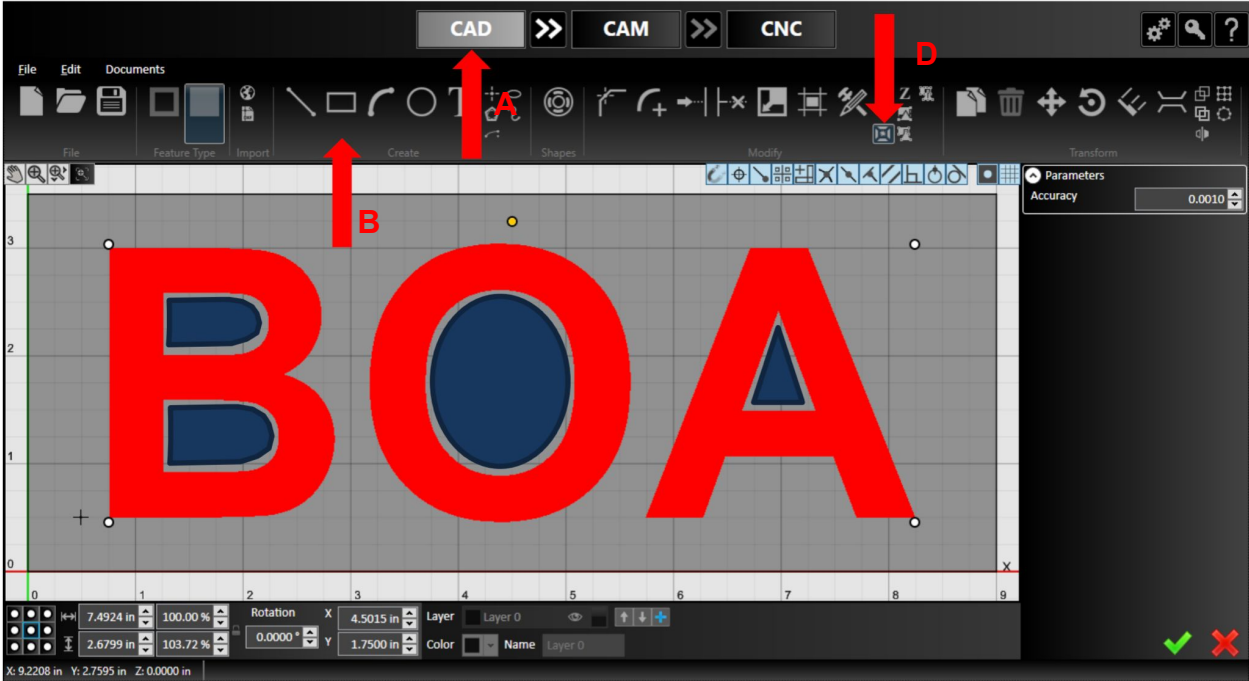
A. Go to CAD
B. Draw the text "BOA" surrounded by a rectangle
C. If you cut this, the centers (shaded in blue for illustration) would fall out
D. Select the text and use the *Explode* button so that it becomes line segments

Bridge Interior Features
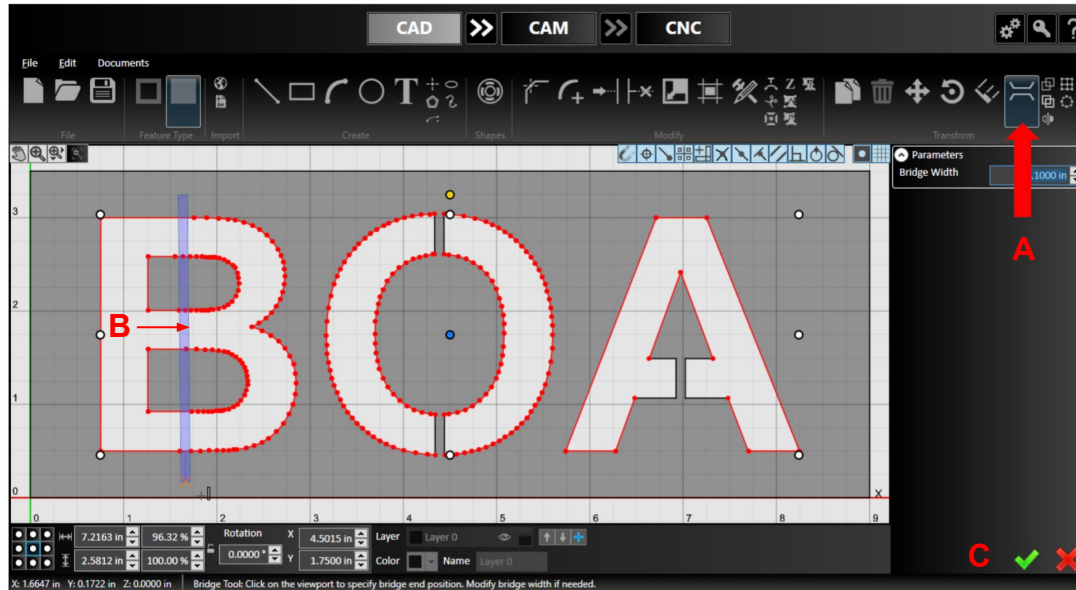
A. Choose the *Bridge* tool
B. Draw bridges to join the interior of the letters to the surrounding box
C. Finalize each bridge by hitting the green *Check* or starting a new bridge
D. You can set the width of each bridge individually and you can move the bridge with your mouse before it is finalized

Create Corner Loops
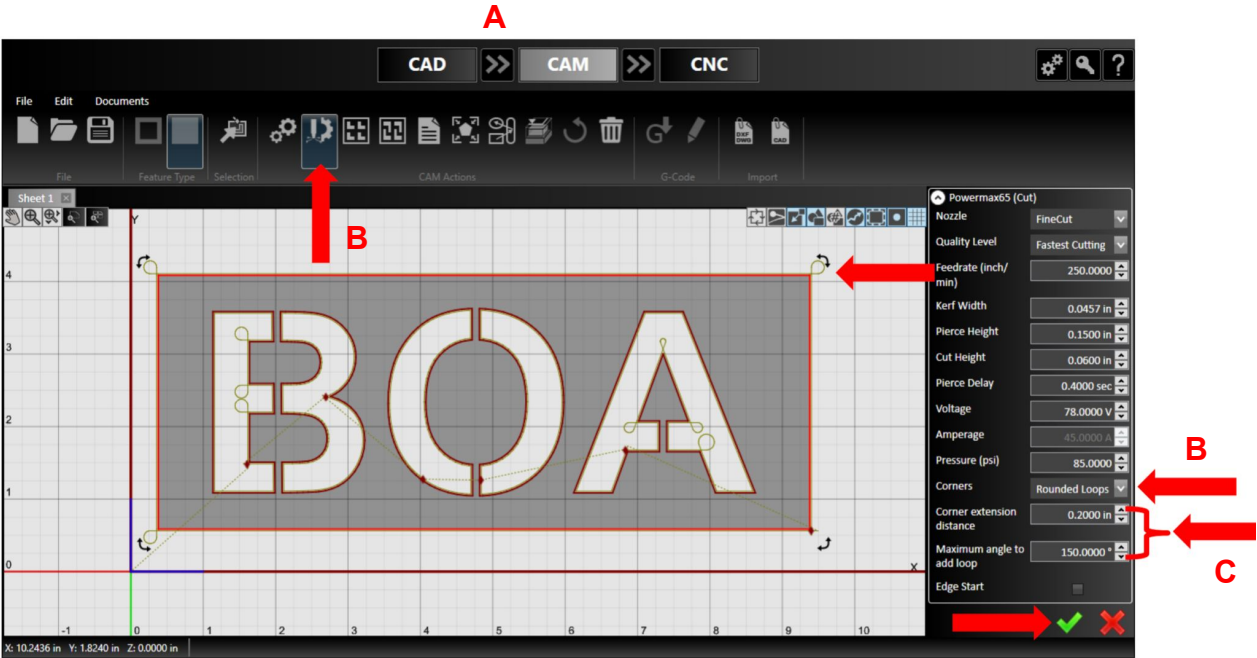
Corner looping minimizes dross on sharp, exterior corners

A. Hit the double arrow to go to CAM mode
B. Go to *Plasma Settings* and set Corners to Round or Triangular Loops
C. Set the Corner Extension Distance; for example 0.15 in
D. Hit the green *Check* and save the CADCAM file

Nesting Multiple Files
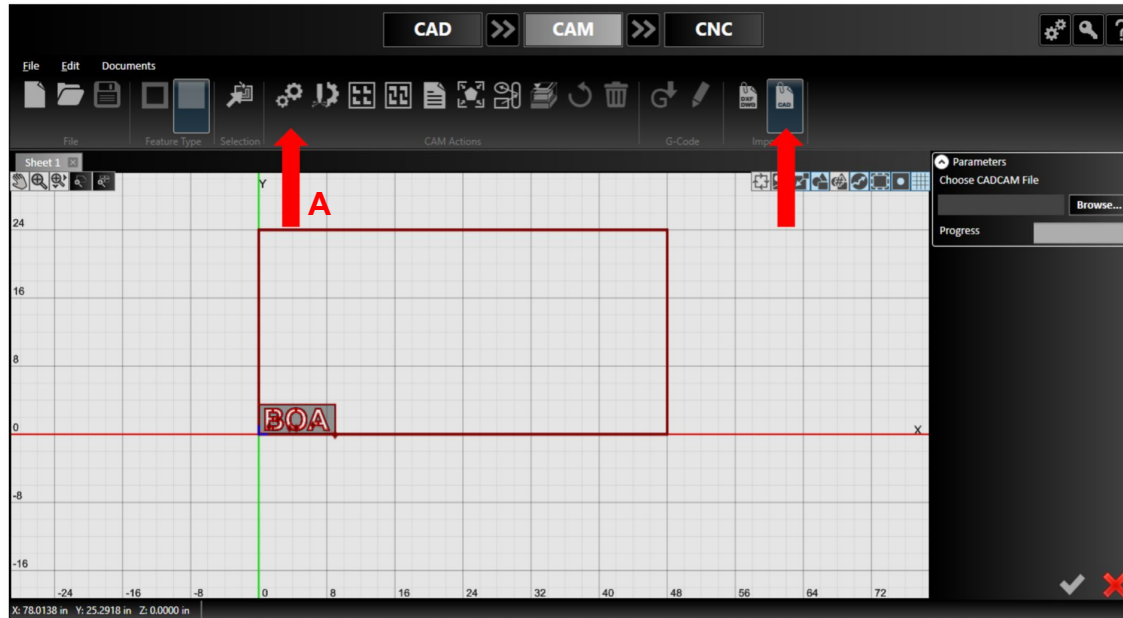
You are not restricted to only nesting the parts in a given CADCAM file; you can reference external CADCAM and DXF files as well

A. Select the *Material Settings* icon
B. 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
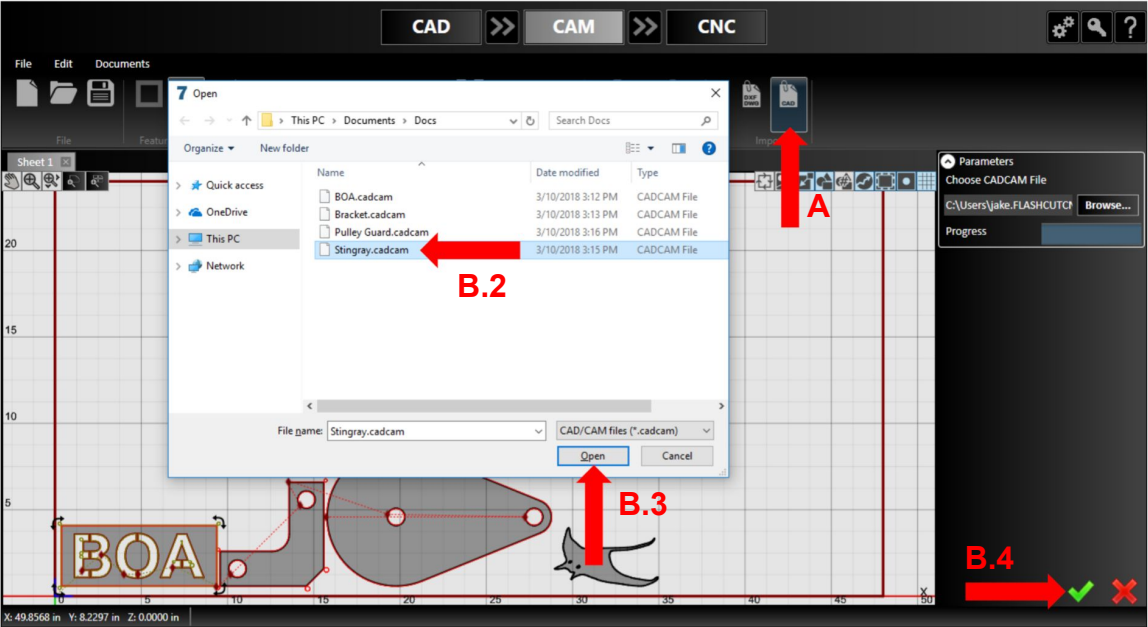
A. Choose the *Import External Files* icon

B. Choose the files that you want to import and then hit the green *Check* for each one

In this case, we are nesting all the files made in the different lessons

Nesting Multiple Files

A. Select the *True-Shape Nesting* icon

B. On the right will be a list of the parts to nest; click on each name and change each count

C. Note that the named part gets highlight in blue when name is clicked

D. Check *Automatically Add Workpieces* if you need more than 1 sheet

E. Fill in the other options such as nesting direction, minimum distances, etc

F. When you are ready, hit the *Start Nesting* button; complex nests will take longer
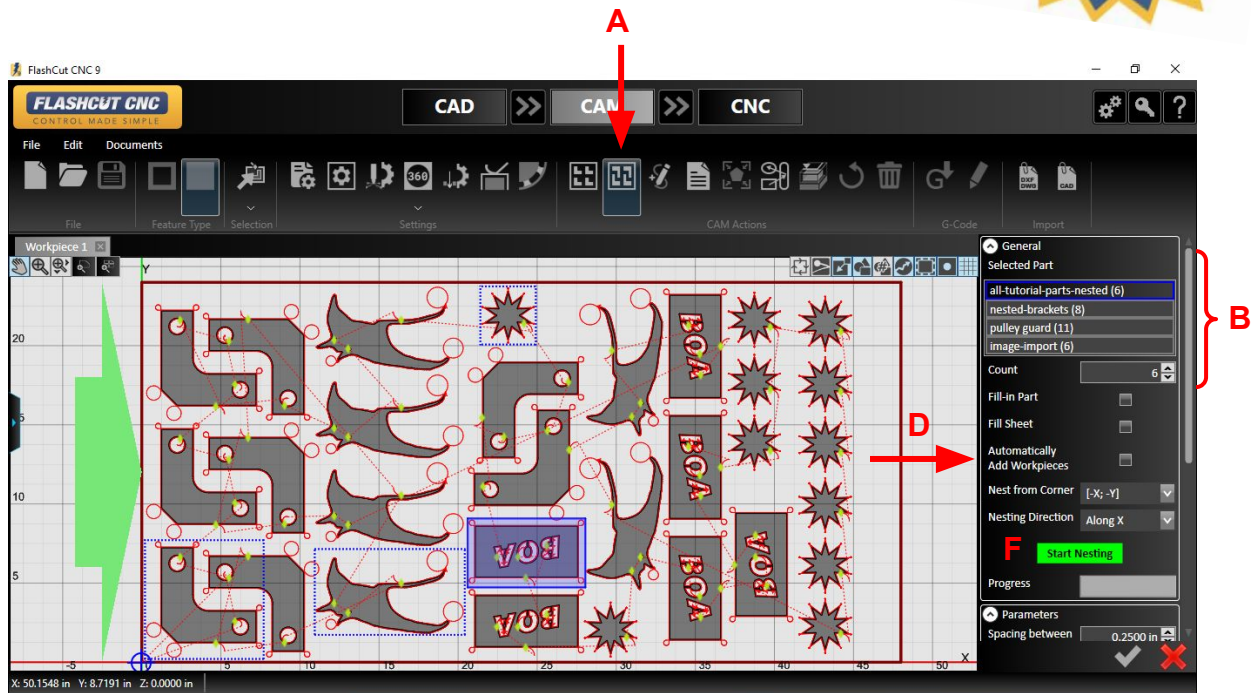
Nesting Report

A. Click on the *Nesting Report* tool to open information about the nest
   a. Information includes number and type of parts and material efficiency
B. Toggle the *Part Numbers* CAM view to see an overlay of the name of each part in the nest
C. Adjust the material price/lb to receive financial information
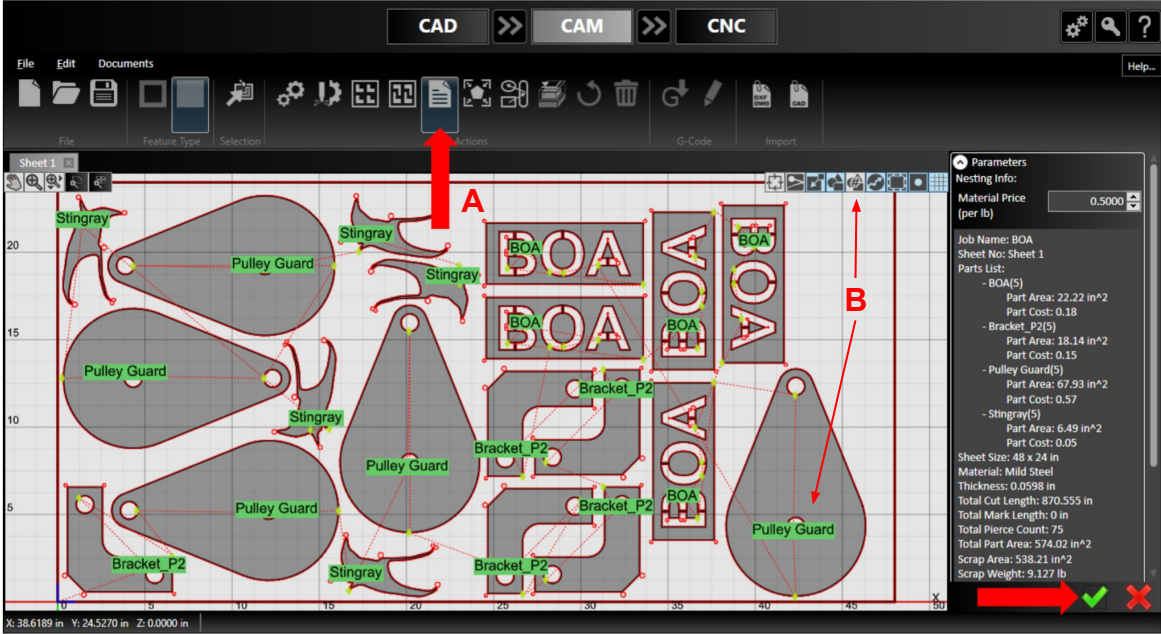
Send Nest to CNC
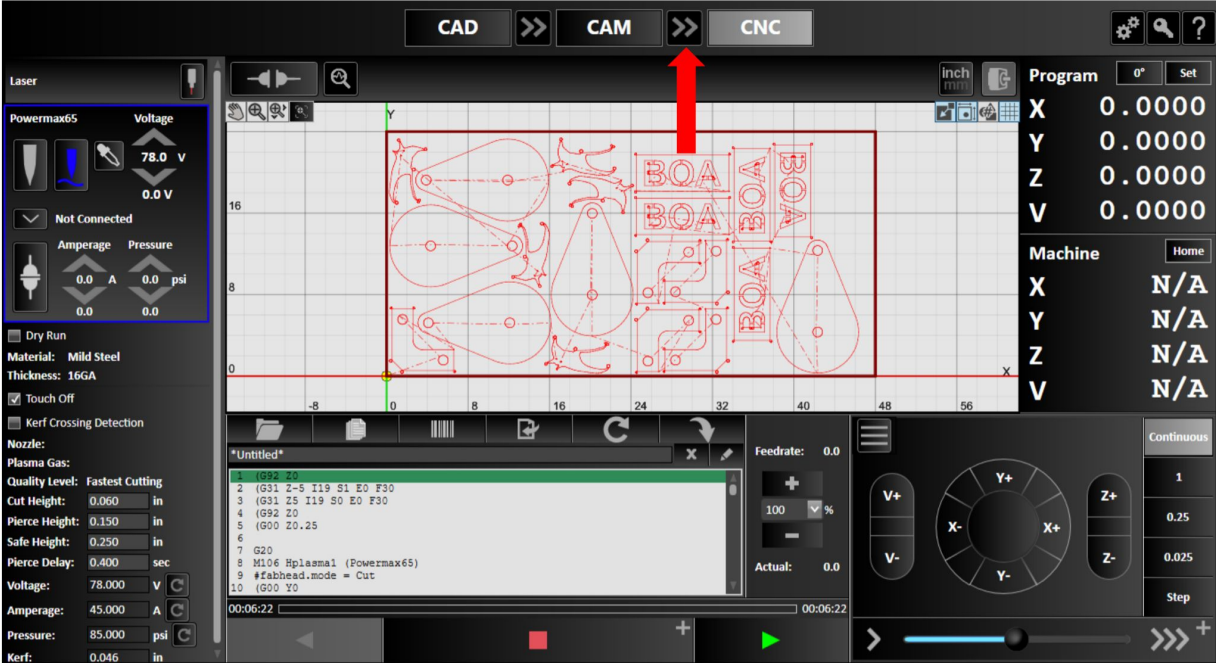
A. Select the sheet that you want to cut

B. Select the double arrow between CAM and CNC
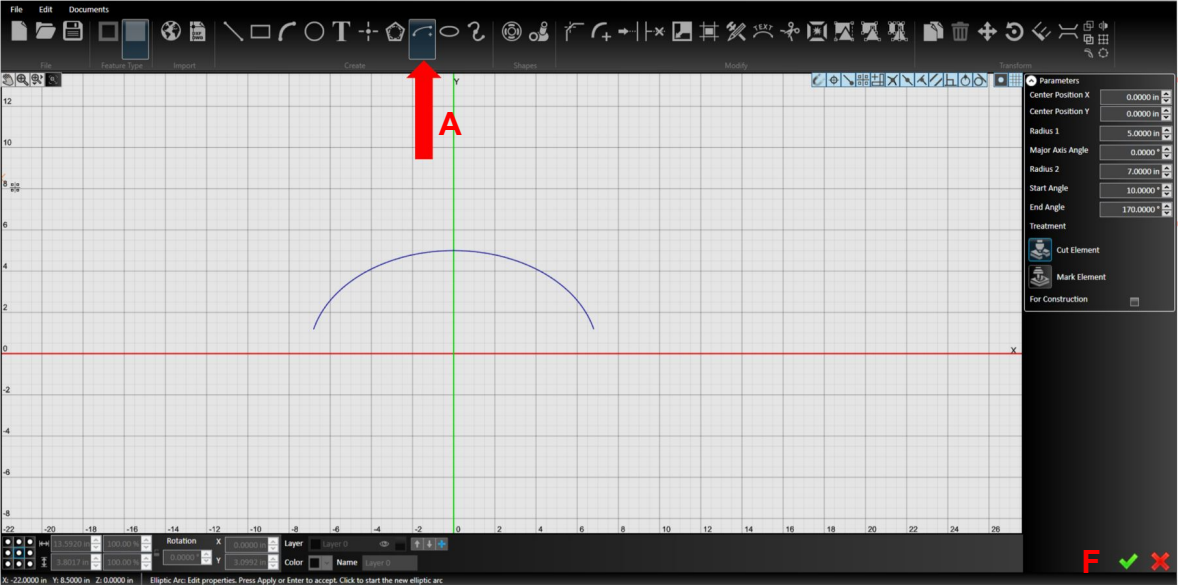
The G-Code of the nested toolpath will be sent automatically to CNC

Lesson 8:
More Advanced CAD Tools

Elliptical Arc

A. In CAD, choose the *Elliptical Arc* tool
B. Chose the center point for the full ellipse with your mouse or by typing on the parameters panel
C. Choose the major and minor radii of the ellipse
D. Choose the start angle and end angle with your mouse or by typing on the right
E. Draw the upper arc only
F. Hit the green *Check*

FLASHCUT CNC
CONTROL MADE SIMPLE

Mirroring
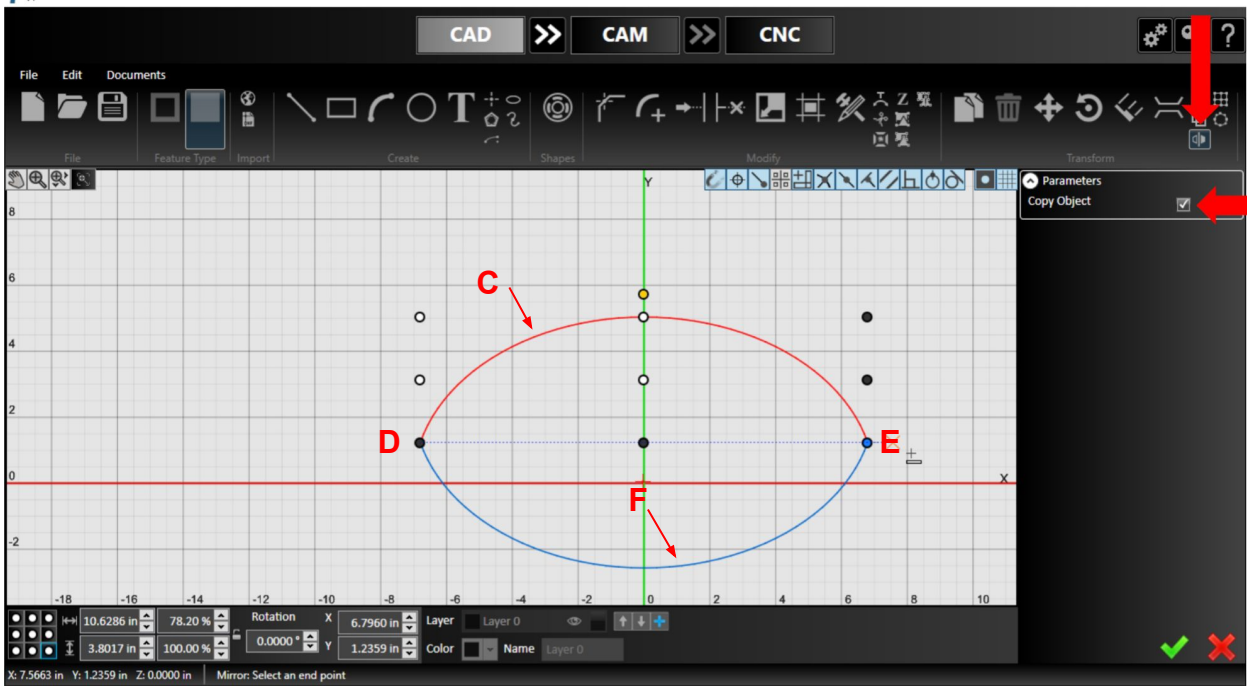
A. Select the *Mirror* tool
B. Check the *Copy Object* box – otherwise, the original copy will be removed
C. Select the elliptical arc to mirror and right click to accept
D. Select the left end of the elliptical arc as one endpoint of the mirror line
E. Select the other endpoint of the mirror line; while you select, the mirror line is dotted
F. The mirrored object will now appear

Curved Text

A. Use the *Text* tool to type the text to curve in any font and size
B. Select the *Curved Text* tool
C. Select the text that you want to curve then right click to confirm
D. Select the curve that you want the text to follow and right click to confirm

Curved Text

PRO

A. Adjust the 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

B. Use the arrows next to the text to move the text along the curve

Note: You can still edit the text in the text editor

FLASHCUT CNC
CONTROL MADE SIMPLE

Shape Welding

A. Explode the text into geometry by selecting the text and hitting the *Explode* button

B. Select all of the geometry that you want to weld together; in this case, it's all of the geometry so you can use ctrl+A or drag over the part to select everything

C. Choose the *Weld Selected Entities* tool to weld everything into one part

Linear Pattern
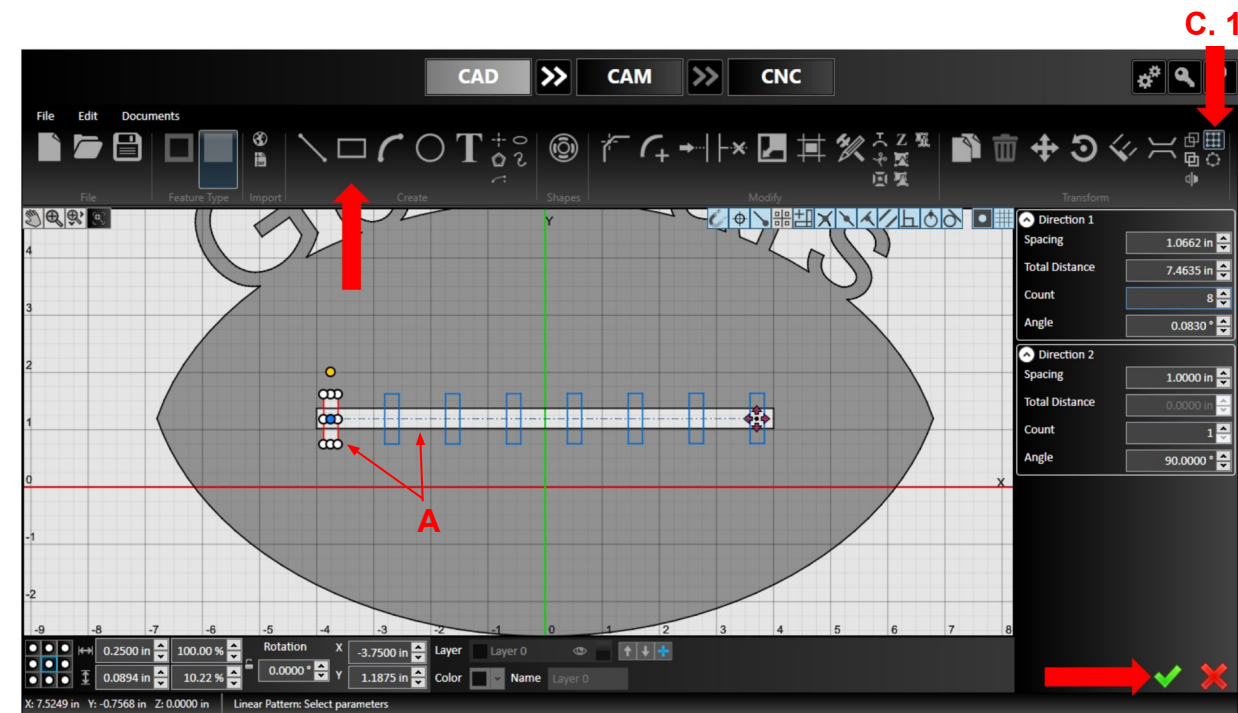
A. Draw a long horizontal rectangle and a short vertical rectangle on the left (see red rectangle in image)

B. Select the red rectangle by alt+clicking any of its sides

C. Click the *Linear Pattern* tool and enter the spacing and count parameters

Note that there is also a *Circular Pattern* tool

Welding and Boolean Operations
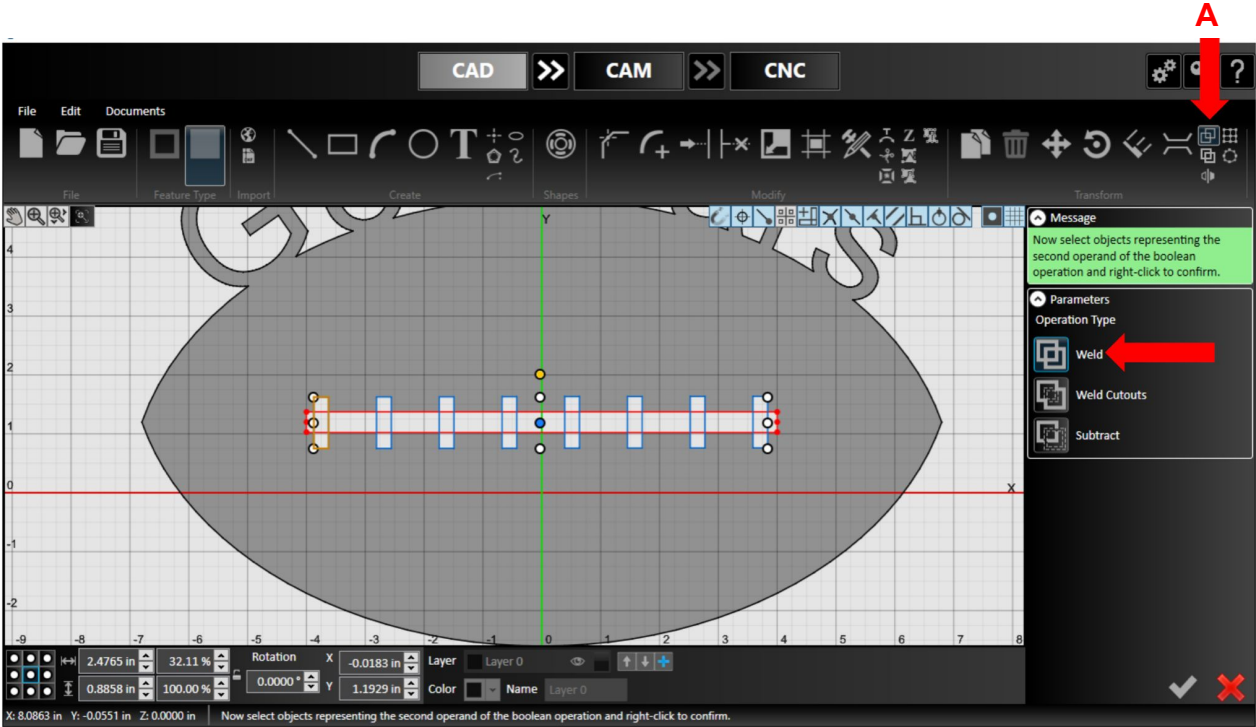
A.   Select the *Weld Entities* tool to weld the objects together once you are happy with the lace pattern

Note: The *Boolean Operations* tool allows you to do welding and other operations such as subtracting entities and welding cutouts

Create the Toolpath

A. Hit the double arrow button to automatically create a toolpath

B. Hit the next double arrow icon to generate a G-Code and send it to the CNC panel
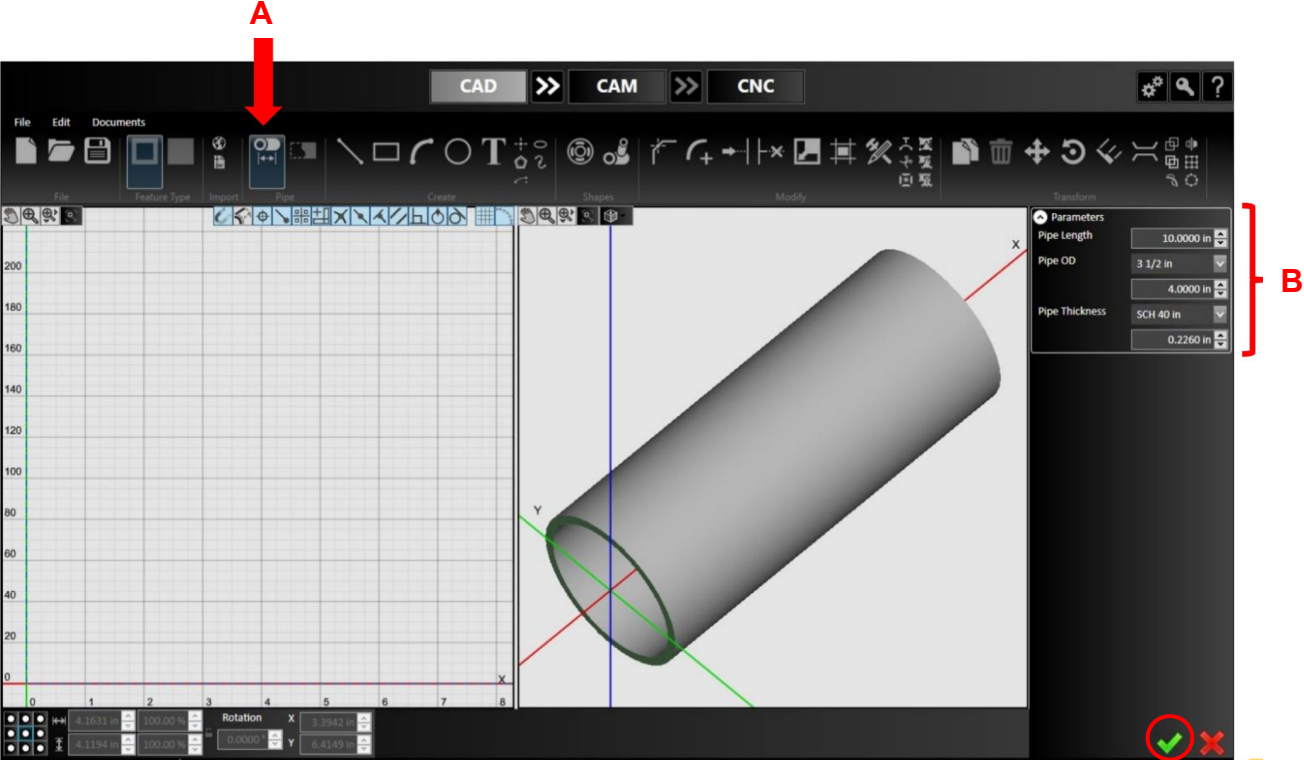
Lesson 9:
Pipe Tools
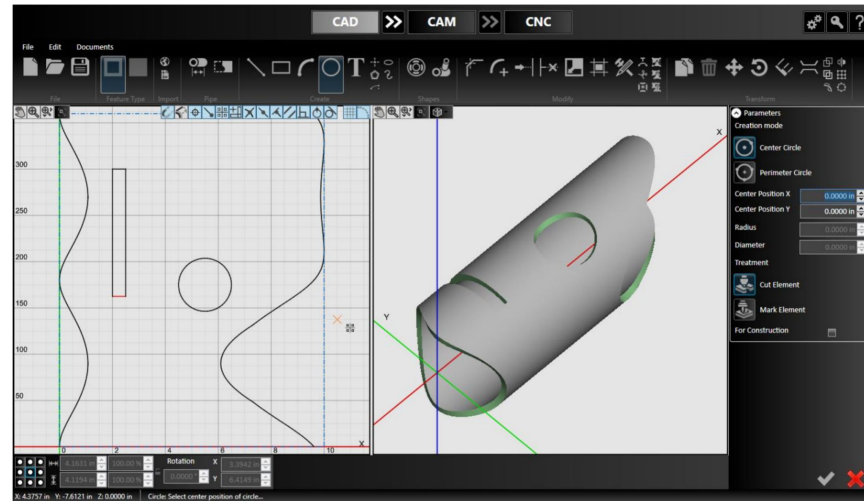
Start New Pipe Drawing
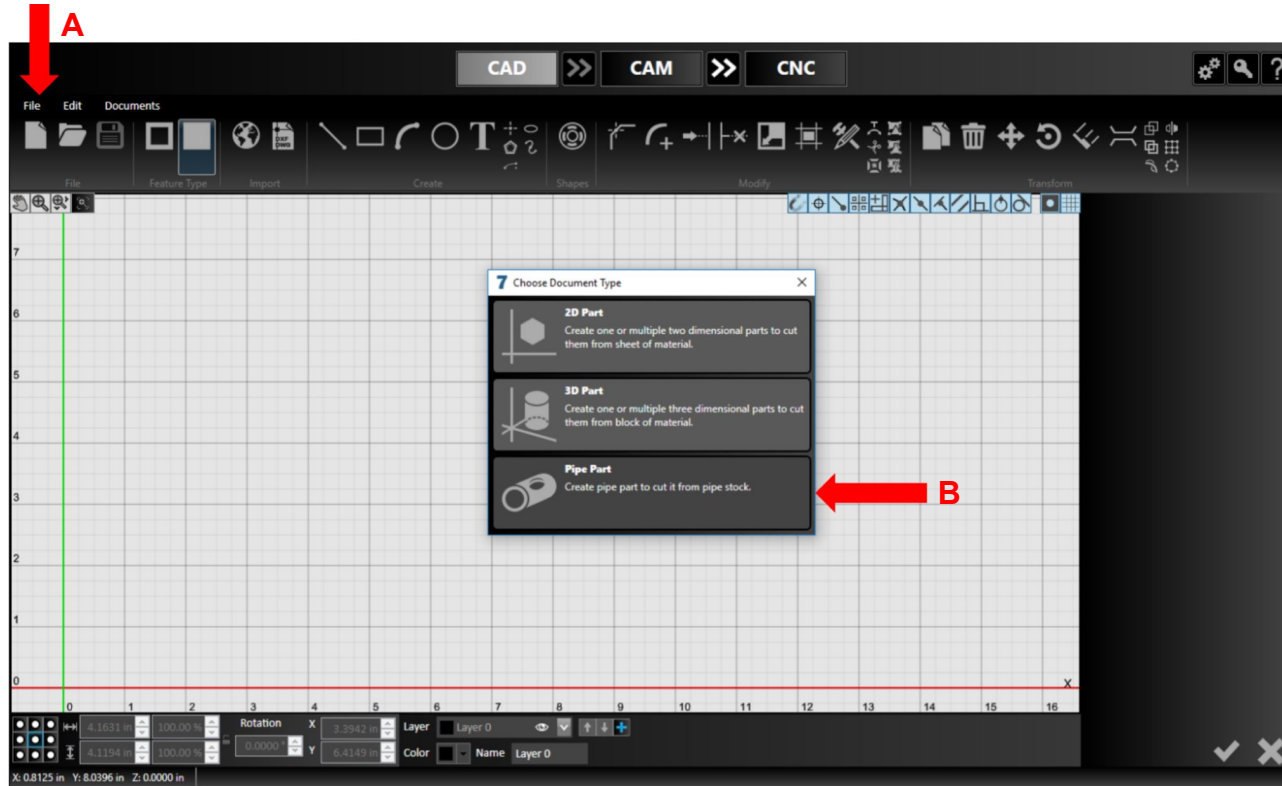
A. Under the File menu choose "New"

B. Choose "Pipe Part"

Choose Pipe Size and Geometry

A. Choose the "Pipe Profile Definition" icon

B. Fill in the appropriate pipe dimensions for length, outer diameter (OD), and wall thickness and check it in

Notice how the Pipe OD and wall thickness can be based on ASTM standards

Determine Mating Conditions
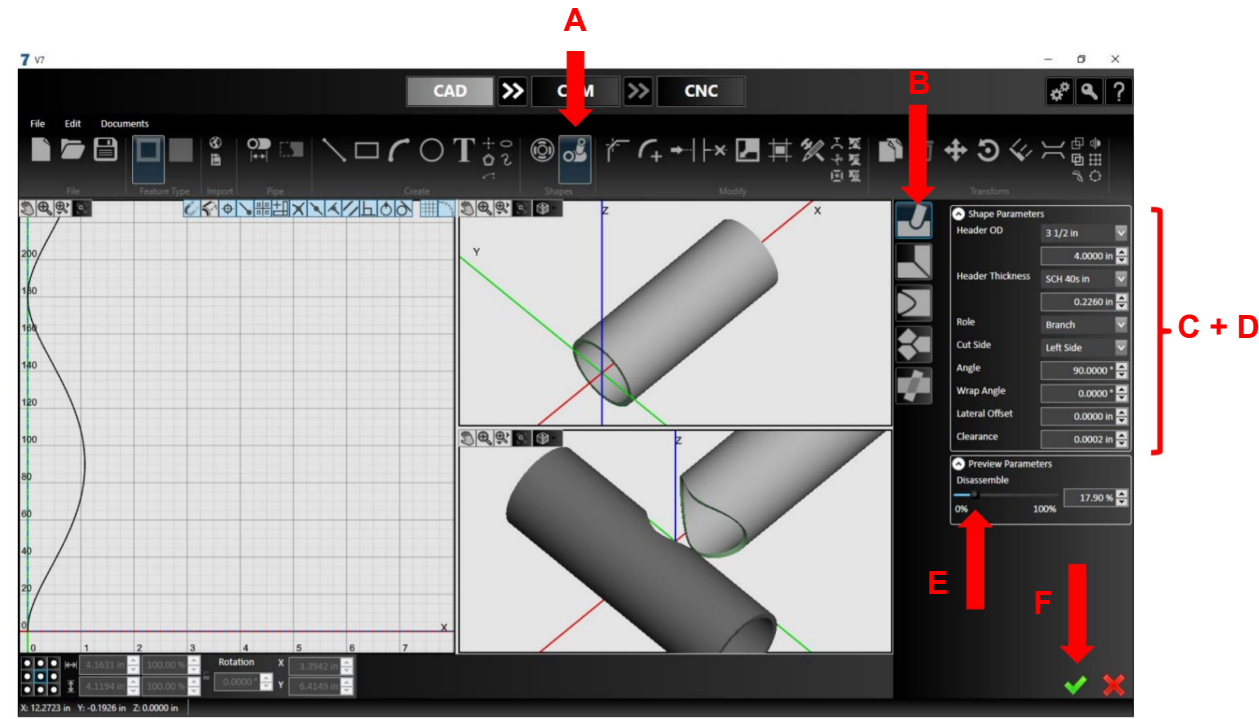
A. Choose the Pipe Joint Shape tool
B. Determine the shape of the joint you are creating – Branch, Header, Miter, End Cap, Wye, or Through Pipe

In this case, the pipe is a 4" branch mating with a header on the left side at 90 deg.

C. Fill in the appropriate dimensions of the mating pipe and which side it is mating with
D. Determine any extra clearance for welding etc.
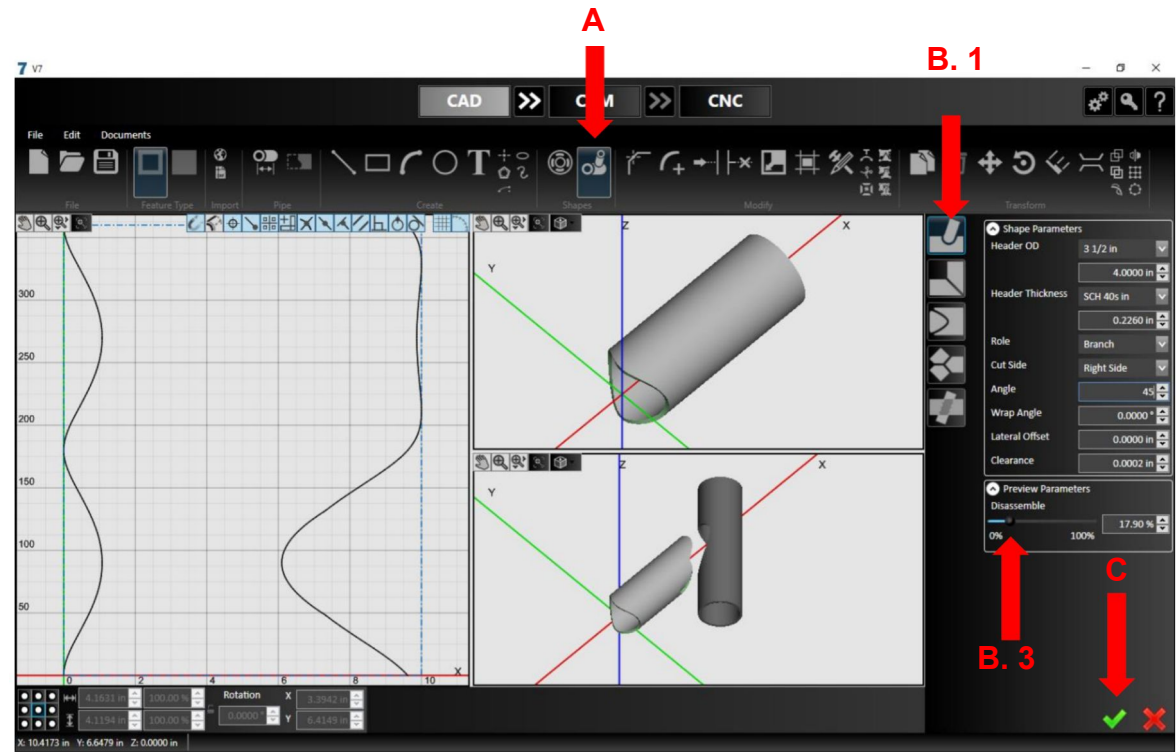E. Preview both part and mating part with disassembly tool
F. Check it in once satisfied

FLASHCUT CNC
CONTROL MADE SIMPLE

New Geometry is Created
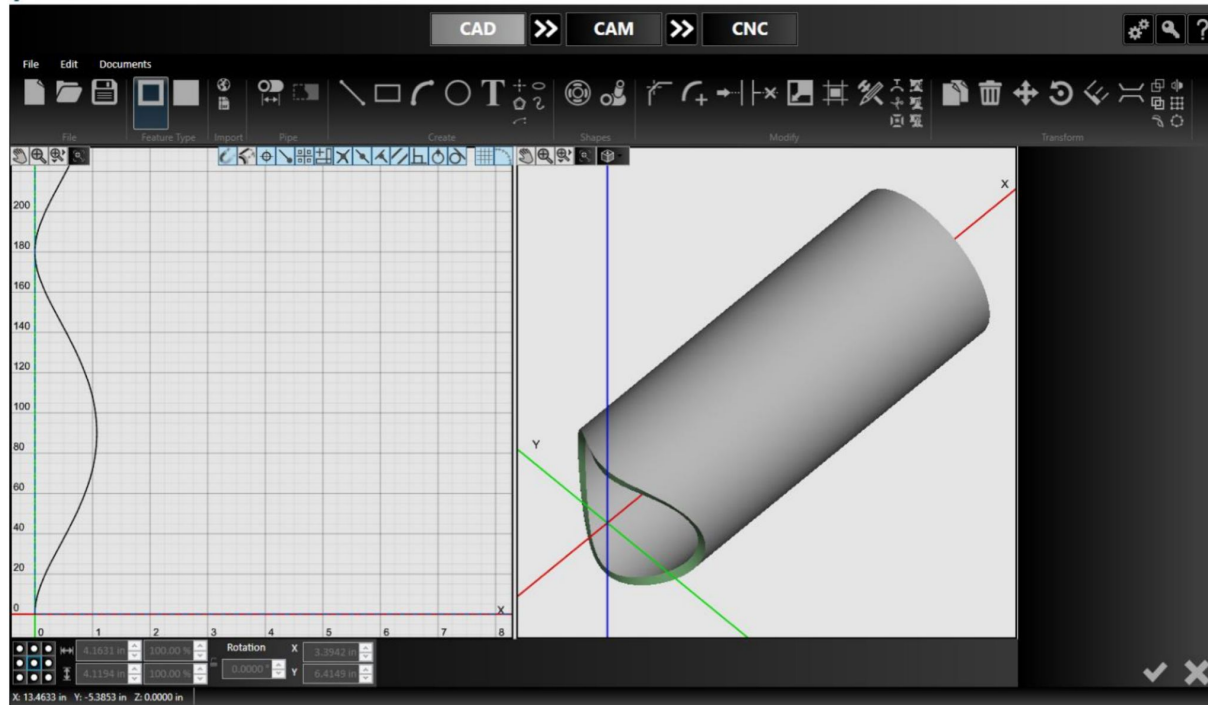
Note:
- The shape tool just created new geometry on the original chosen pipe
- An unwrapped 2D view on the left and a solid 3D view on the right are shown
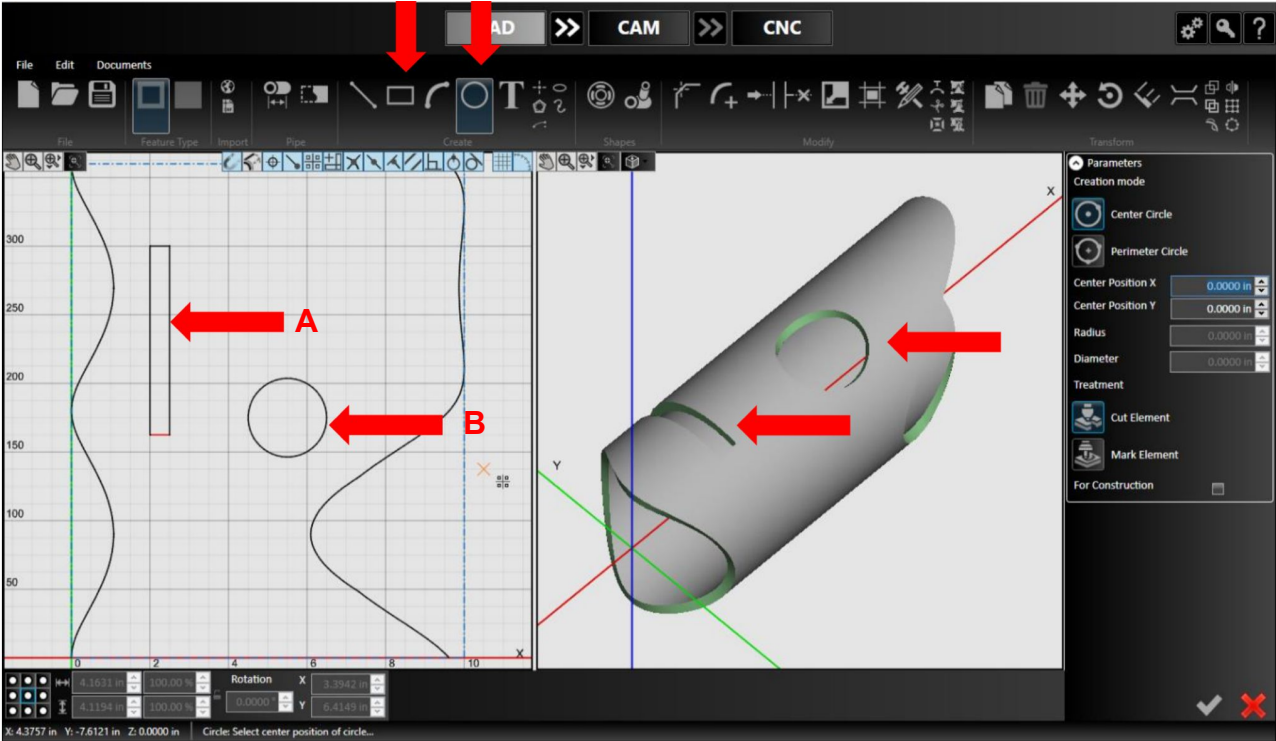
Add the Other Shape Parameters

A. Choose the Pipe Joint Shape tool again

B. Determine the shape of the joint you are creating

In this case, we will make the right side of a 4" pipe mating at 45 deg.

C. Check it in

Draw more Features in 2D Window
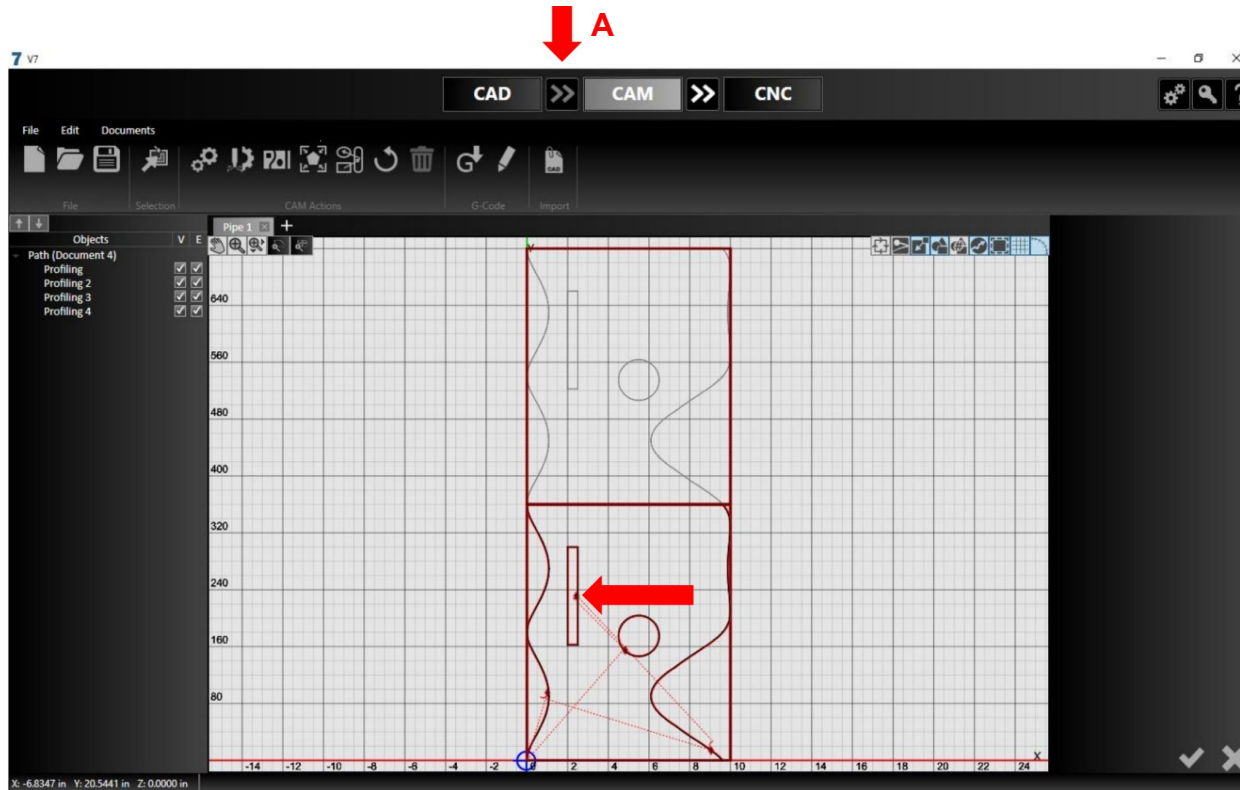

A. Draw a rectangle in the 2D unwrapped view

Notice how it becomes a slot in the 3D.

B. Draw a circle
C. Draw any shape you want

Note that Text needs to be exploded. The Bridge tool and Boolean tools also work in pipe mode.
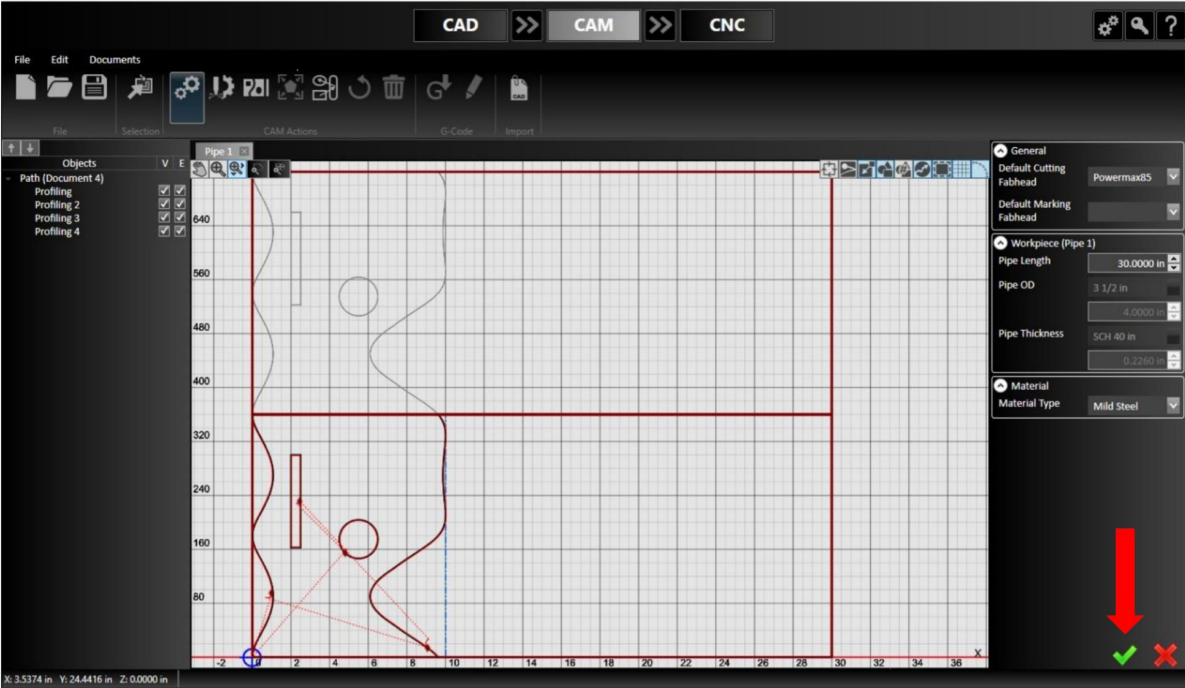
Send the Pipe to CAM

A. Choose the double arrows or the CAM button

A toolpath is created with using the original pipe dimensions

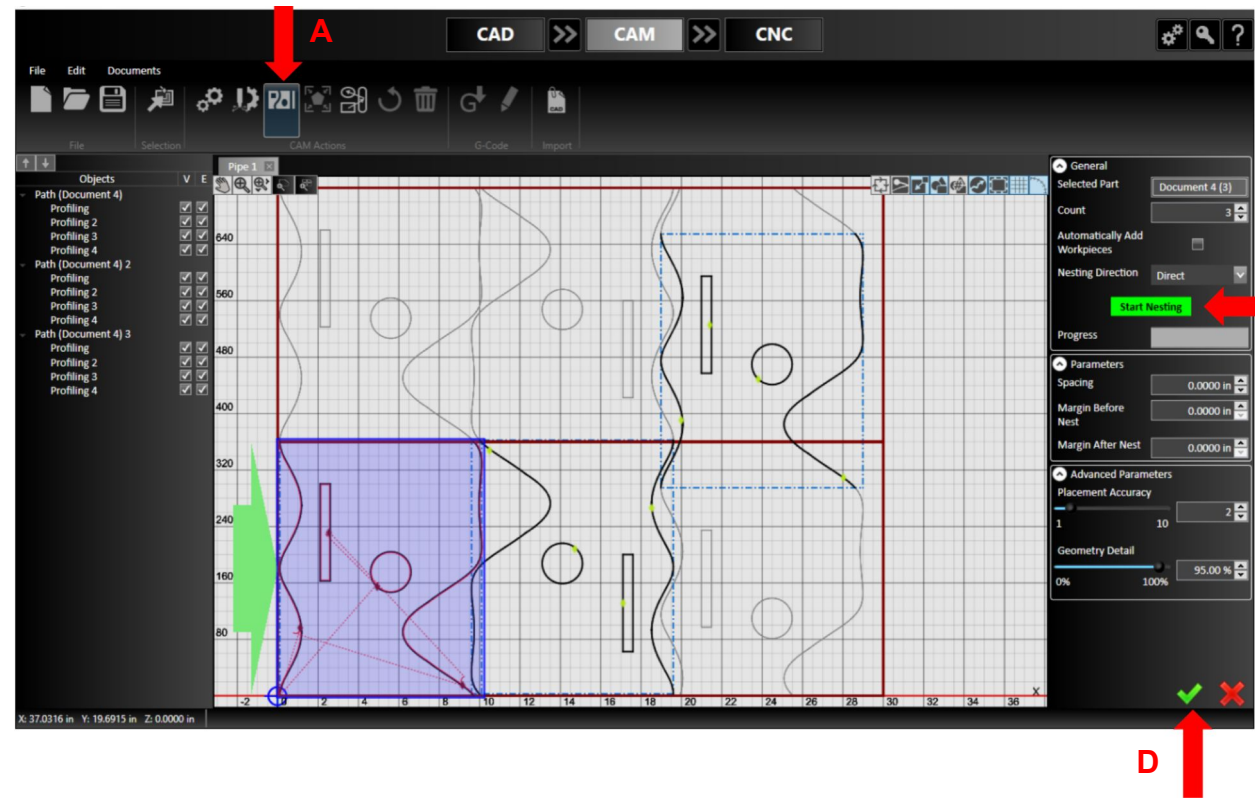The red denotes the main toolpath. The gray is a shadow view of the same toolpath.

Note: The lead-ins, lead-outs can be manipulated identically to how they are edited in 2D

Enter the Length of the Stock Pipe

A. Change the pipe length in CAM to 30"

Nest the Pipe
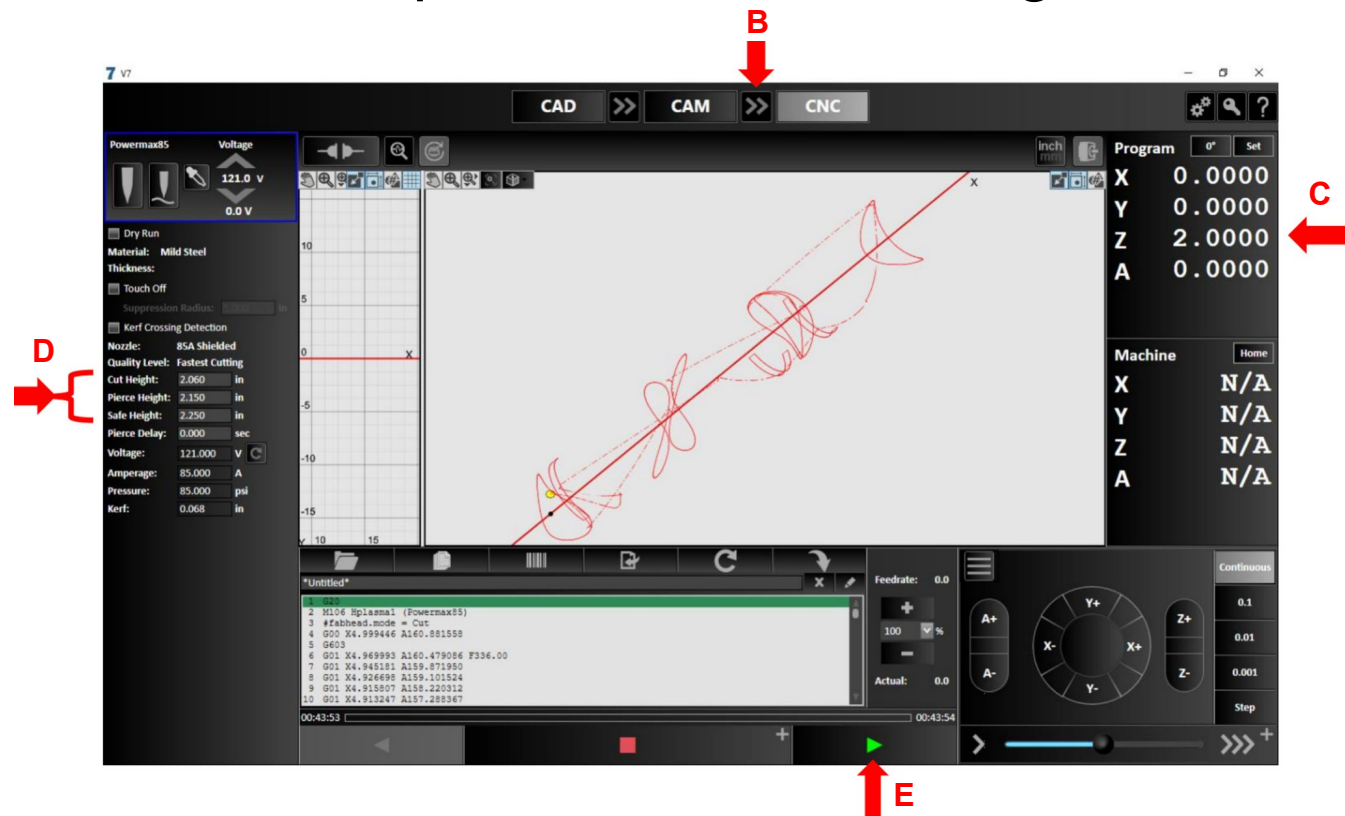

A. Choose the Pipe Nesting Icon

B. Change the Count to 3

C. Hit the Start Nesting button

Notice how the parts flipped and staggered to create an efficient Pipe Nest
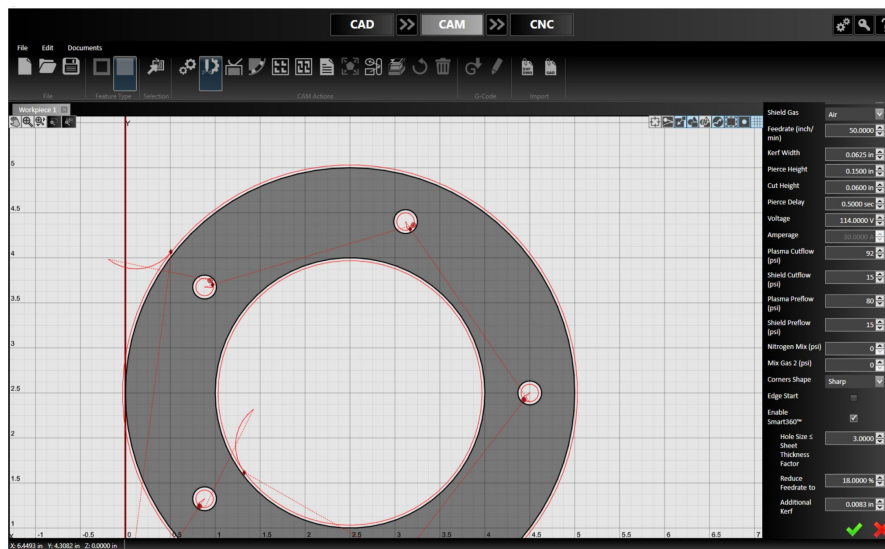
D. Check it in

Send the Pipe to CNC for Cutting

A. Make sure your configuration is properly set for rotary pipe cutting
B. Hit the double arrows to create the G-Code and send the toolpath to CNC
C. Set the surface of the part to the radius of the pipe
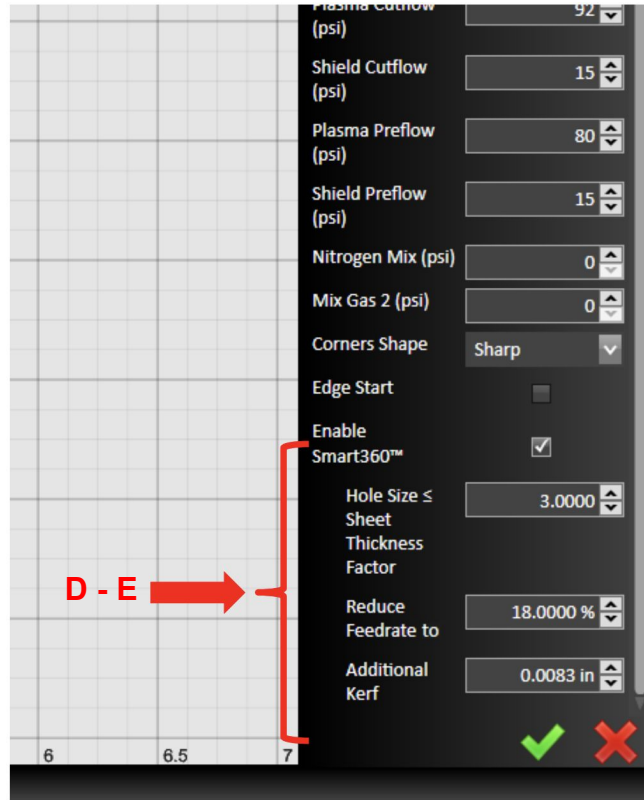D. Make sure the Cut Height, Pierce Height and Safe Height are properly set
E. Run the part

Lesson 10: Smart360™

Enable and Configure Parameters

A. Make sure you have a compatible torch configured
B. Open Plasma Settings in the CAM module
C. Check the "Enable Smart360™" toggle
D. Enter the size of the holes you want to apply Smart360™ to, based on the sheet thickness
E. Change the feedrate reduction factor as desired (default is 18%)

If the selected parameters are compatible, an Additional Kerf value will be generated. Otherwise, the user can enter a value based on experience

FLASHCUT CNC
CONTROL MADE SIMPLE
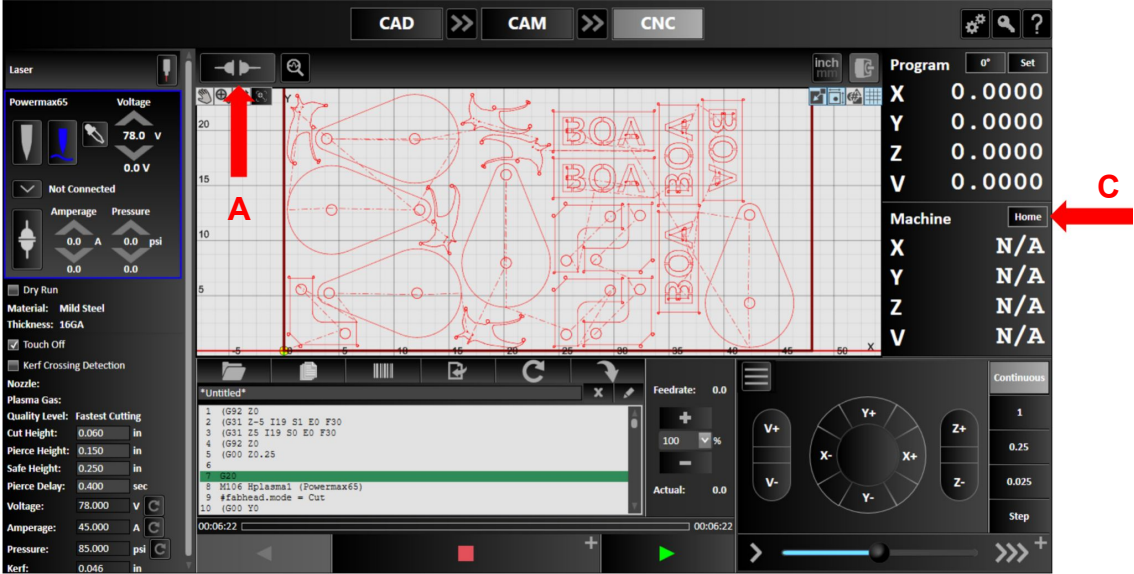
Modify Lead-In
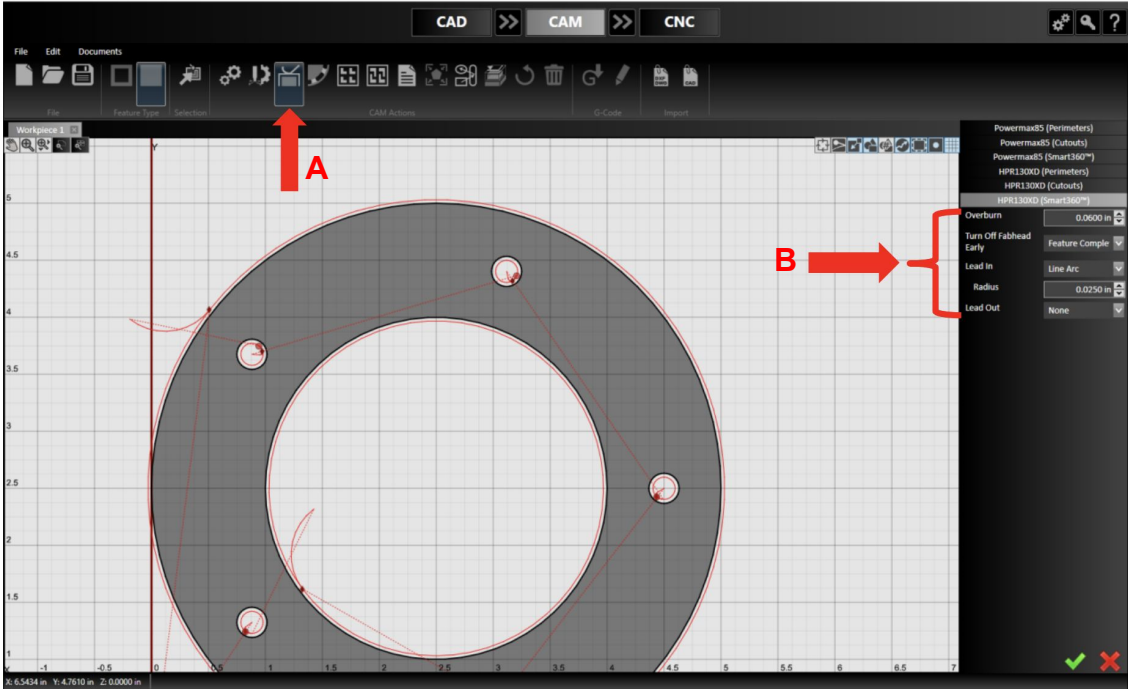
A. Open the Breaks tool to change lead-in parameters

Each aspect of the Smart360™ lead-in is generated automatically

B. For advanced users, each parameter can be individually modified

Lesson 11:
Cutting a Part in CNC

Activating Controller License

A. Make sure the controller is connected by selecting the Connect Button
B. Agree to the "Safety First" message and hit OK to upgrade the firmware
C. If you configured homing, selecting the *Home All;* the machine will seek the switched on all axes and set all of the Machine Coordinates to 0

Set your Program to Zero
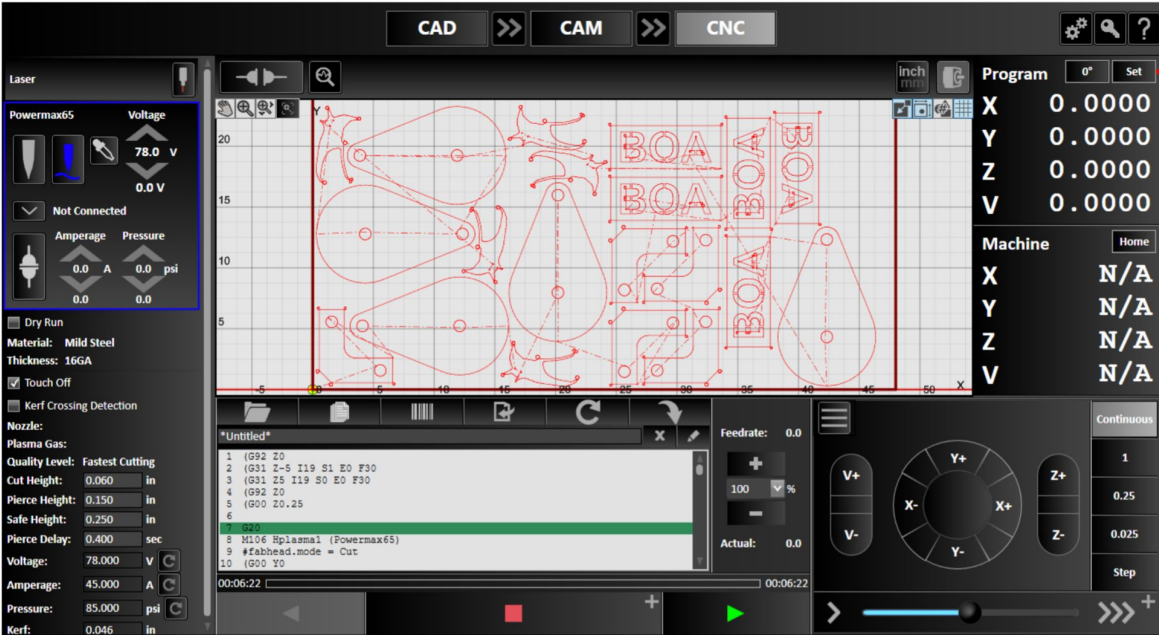
A. Jog to the zero point of your program by clicking on any of the jog buttons either in *Continuous* mode or any of the discrete distances; the slider bar determines speed

B. Set the X & Y Program Coordinates to zero with the *Set* button and *Zero X* and *Y*

C. If you have a sensor on your Z axis, set your Z axis coordinate by clicking *Set* and *Sense Z Zero;* if you don't have one, select *New*… under *Set* to set your coordinates

Cut the Part
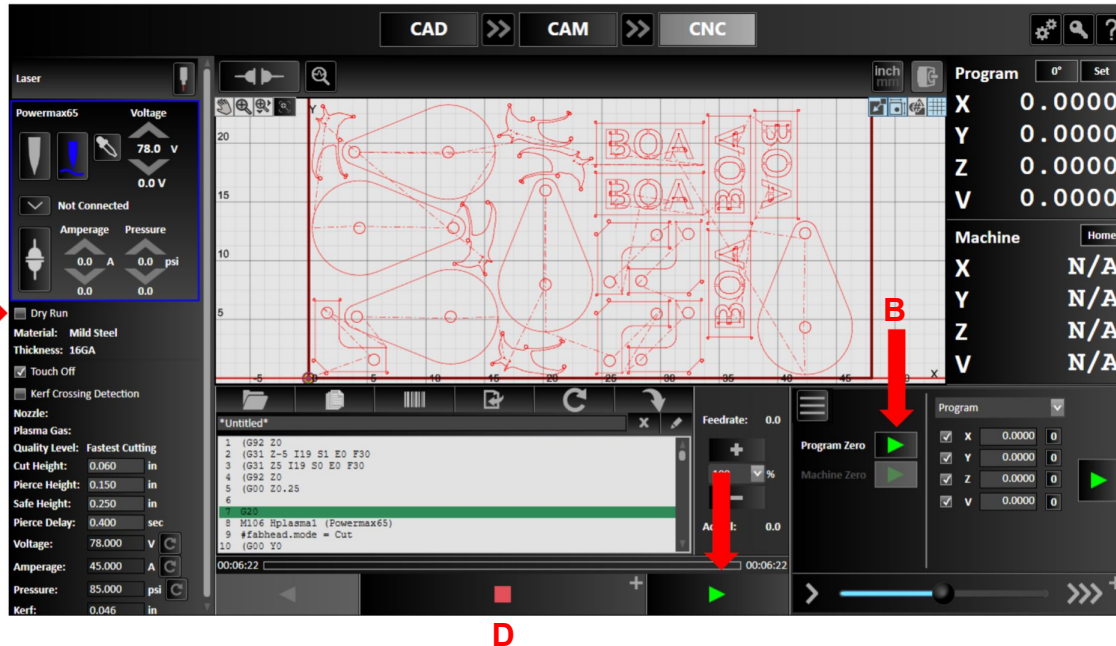
A. Verify the toolpath by clicking *Dry Run* and then the green *Run G-Code* button

B. If everything looks good, you can move back to program zero by switching from Job Mode to Point Mode and selecting the green *Program Zero* button

C. To cut the part, uncheck *Dry Run* and select the green *Run G-Code* button

D. You can stop the program at any time by choosing the red *Feed Hold* button

Program Flow Control
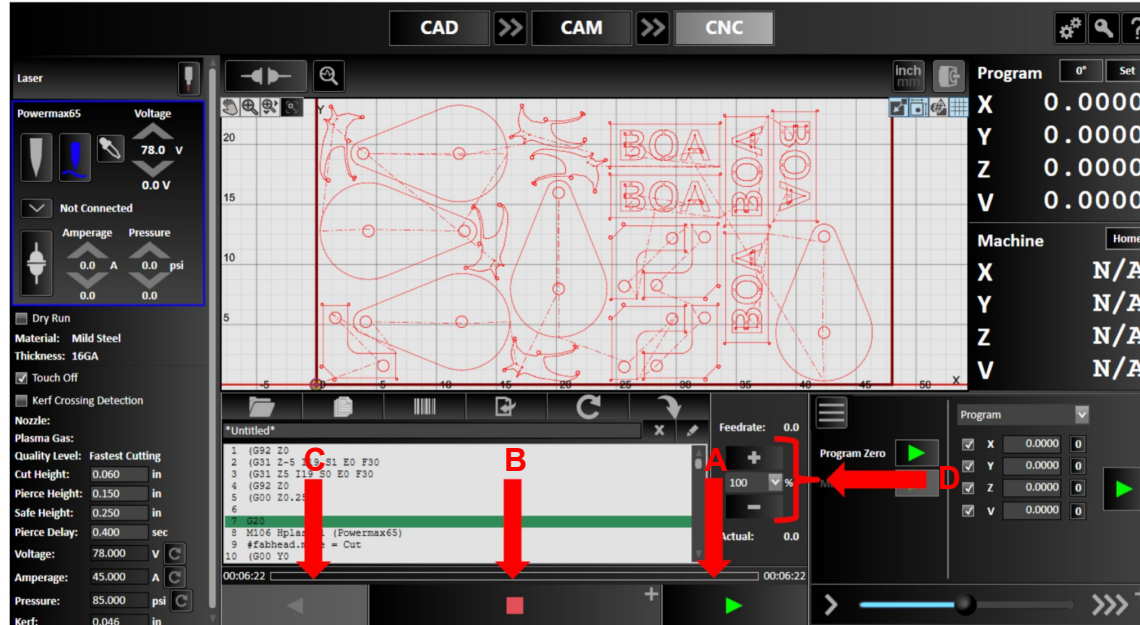
A. The *Run G-Code* buttons runs the program forward and cuts the parts

B. The red *Feed Hold* button ramps down the motors and stops the program

C. The gray *Run G-Code in Reverse* button will run backwards, but won't cut

D. You can speed up or slow down the feedrate in real-time by hitting the + or - feedrate buttons

Jump to Line
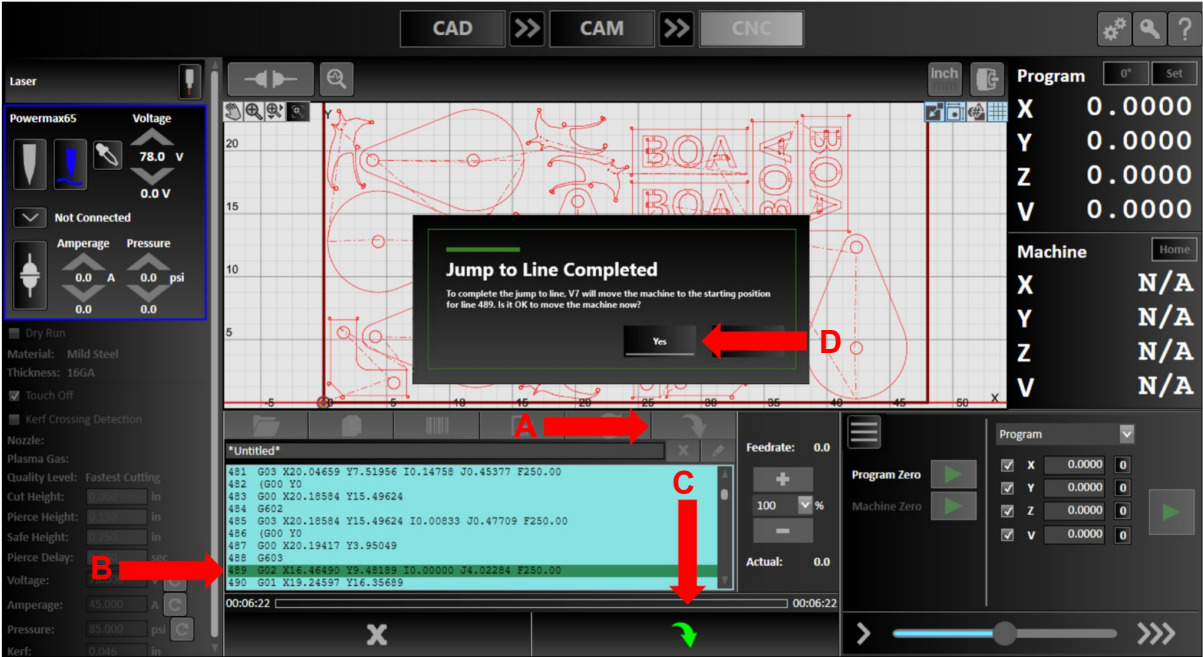
A. Jump to any line of the program while idle by pressing the *Jump to Line* button

B. You can now select any line in the Program Listing Boc or any command in the Viewport

C. Click the *Execute Jump To Line* button

D. A dialog will pop up verifying the Jump to Line was Completed; choose *Yes* to move the machine to the beginning of the line

You can run the program forward or backwards using normal program flow control
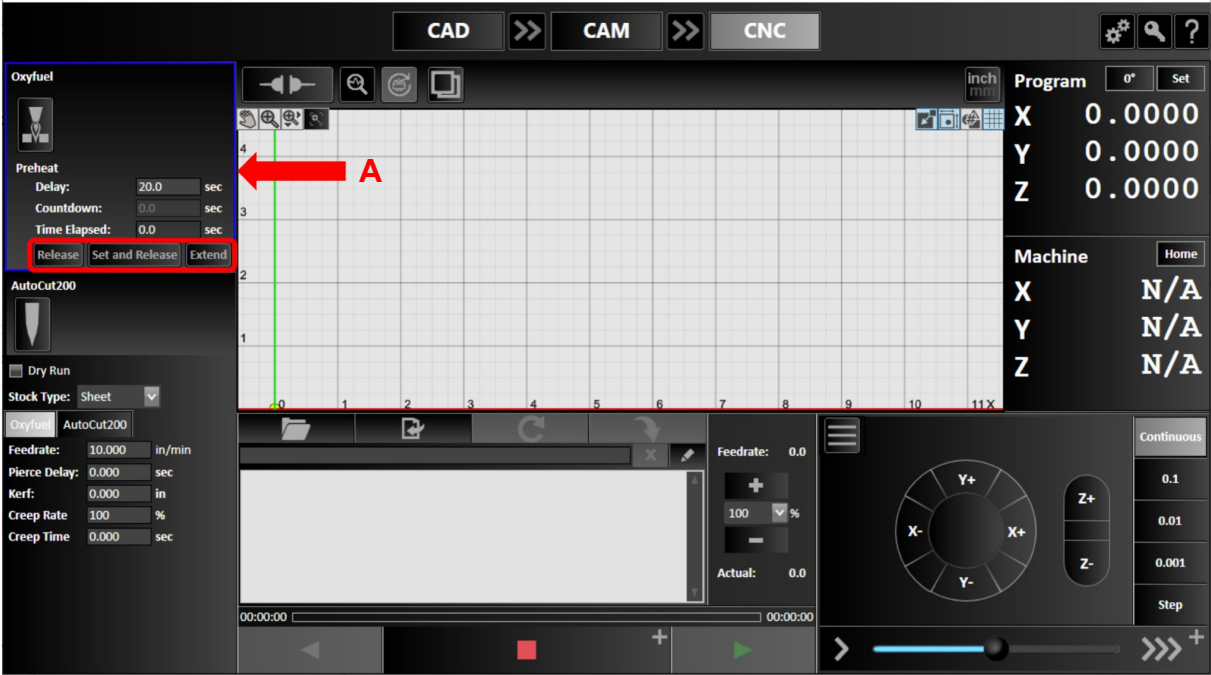
Progress Meter
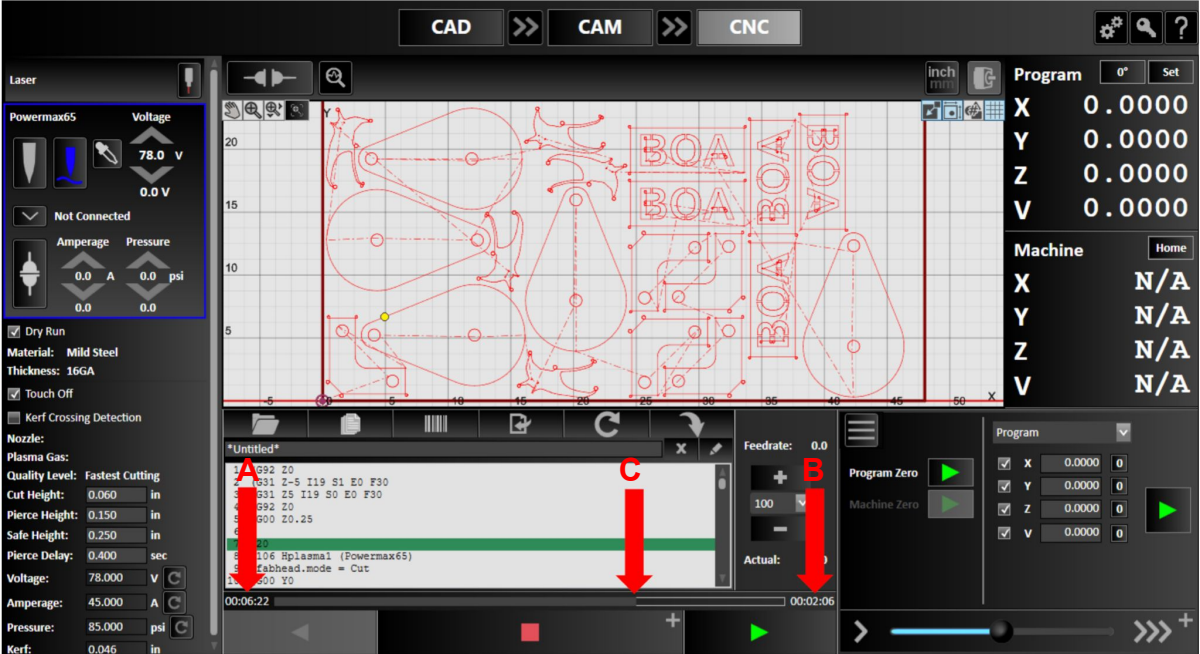
Automatically calculate program runtime

A. The value on the left shows total runtime
B. The value on the right shows time remaining
C. The meter fills as the program runs to visually indicate progress
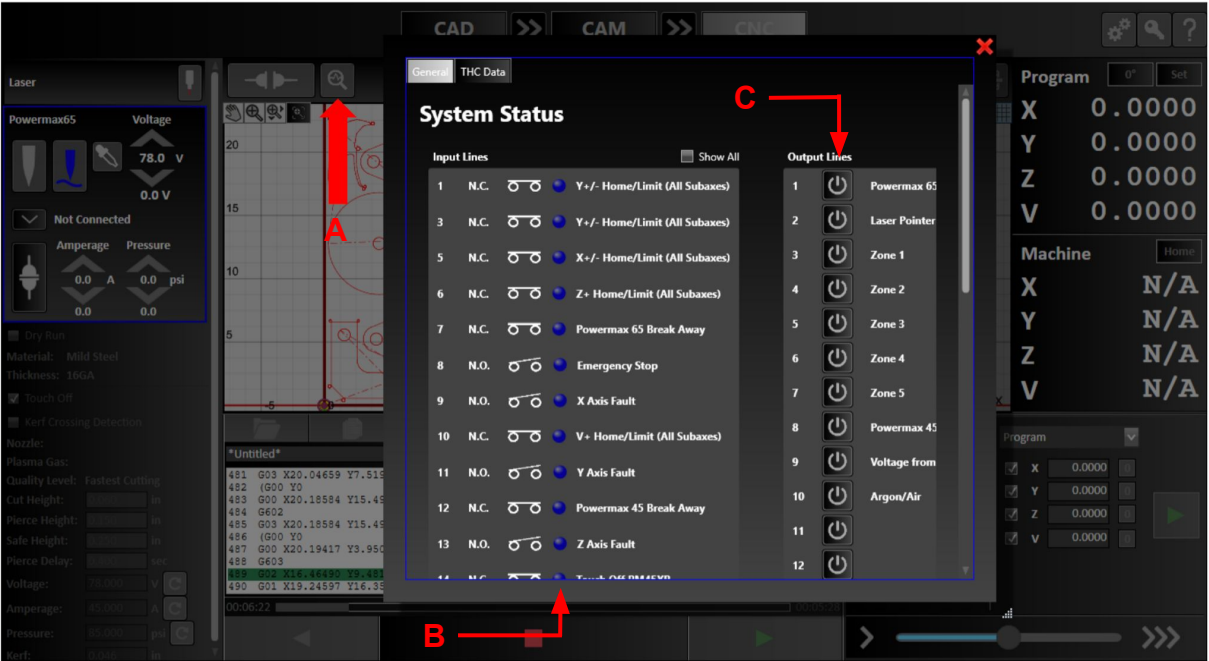
Advanced Oxy Fuel Control

Dynamically release oxy fuel during preheat process

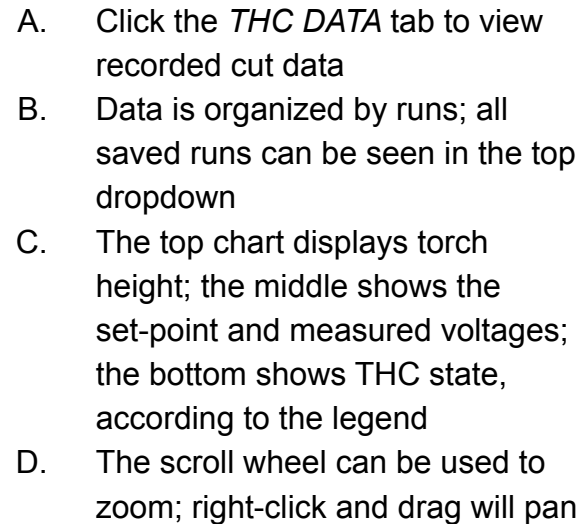A. Choose to overwrite the delay with the current time

Lesson 12:
Diagnostics & Help

Diagnostics & Help

A. The *System Status* panel is accessible through the Magnifying glass icon

B. Live status of *Input Lines* is shown by the switch icons and blue LEDs (only active if the software is connected to the controller)

C. Live status of the *Output Lines* are also available; turn any output on and off simply by clicking on its button

THC History

A. Click the *THC DATA* tab to view recorded cut data
B. Data is organized by runs; all saved runs can be seen in the top dropdown
C. The top chart displays torch height; the middle shows the set-point and measured voltages; the bottom shows THC state, according to the legend
D. The scroll wheel can be used to zoom; right-click and drag will pan

Help Screen and User's Guide

A. The *Help* screen can be accessed by clicking on the *?* button

This shows the serial number as well as current software and firmware versions

B. *Build Support File* will create a ZIP with configuration settings and a log of past commands; this can be emailed to our support staff for in-depth diagnostics

Any online manuals are available through the *Help* window in some versions
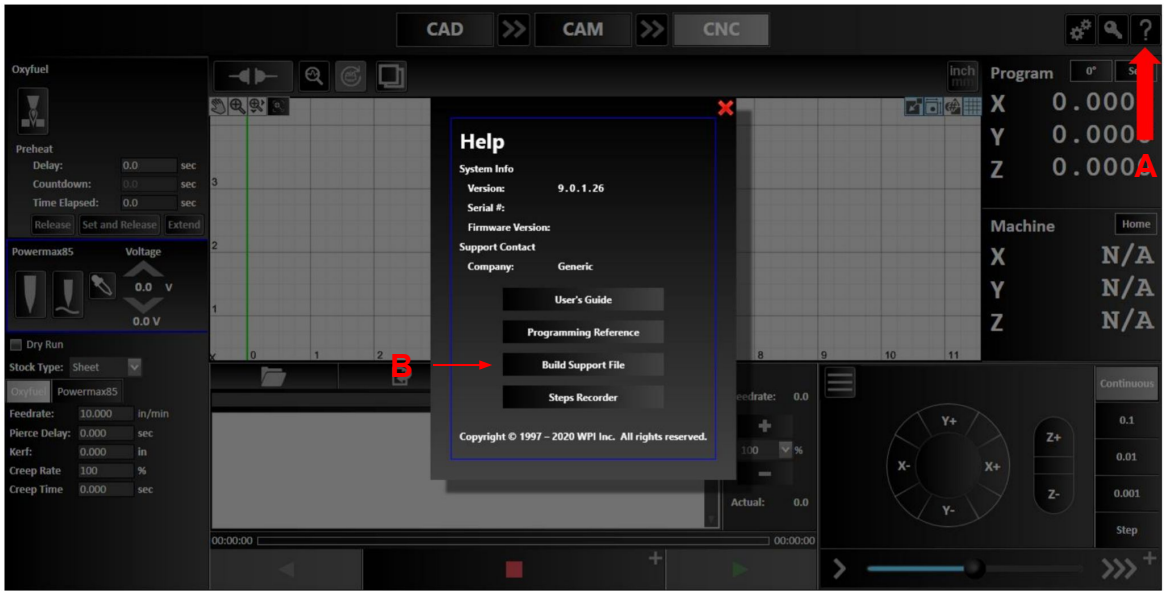
User's Guide
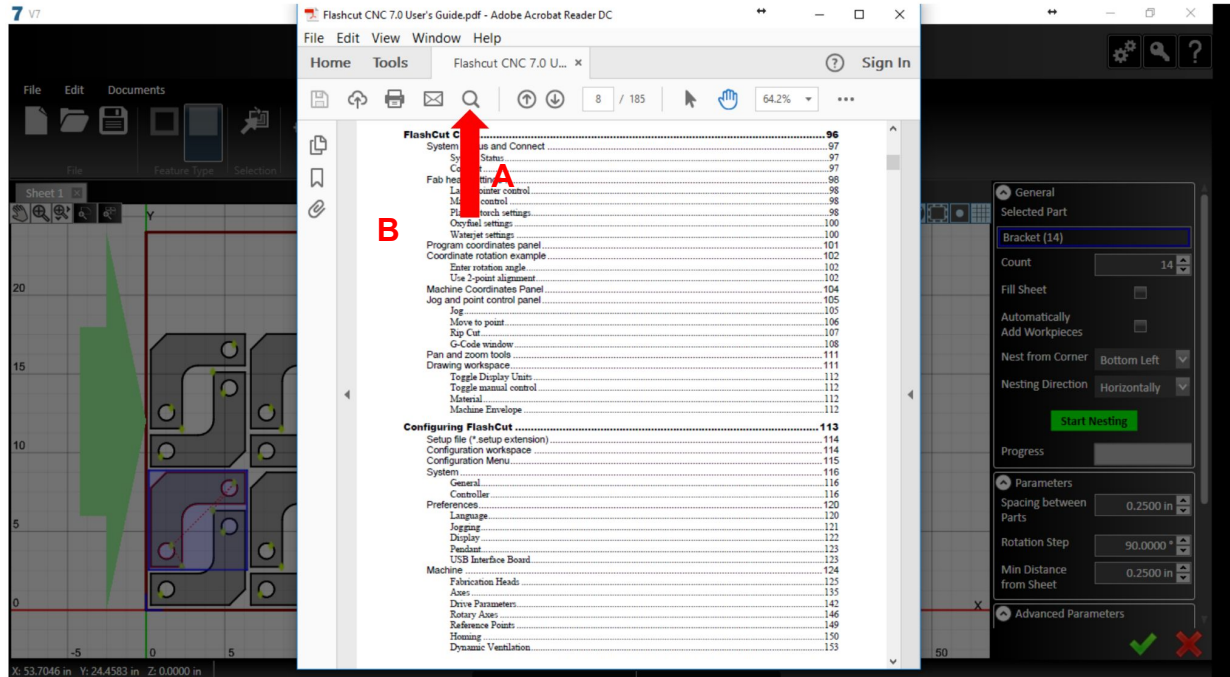
A. The User's Guide will launch on the computer's default PDF application

B. Clicking on any of the sections in the Table of Contents will bring the user to the chosen page

C. The Find function can be used to locate relevant information

Five Languages Supported

A. To change a language go to Configuration and select *Language*
B. Choose the language you need from the drop down menu
C. You will need to save the configuration, exit, and re-enter the program for the language you register

Note: Only the software will be changed; all manuals are in English
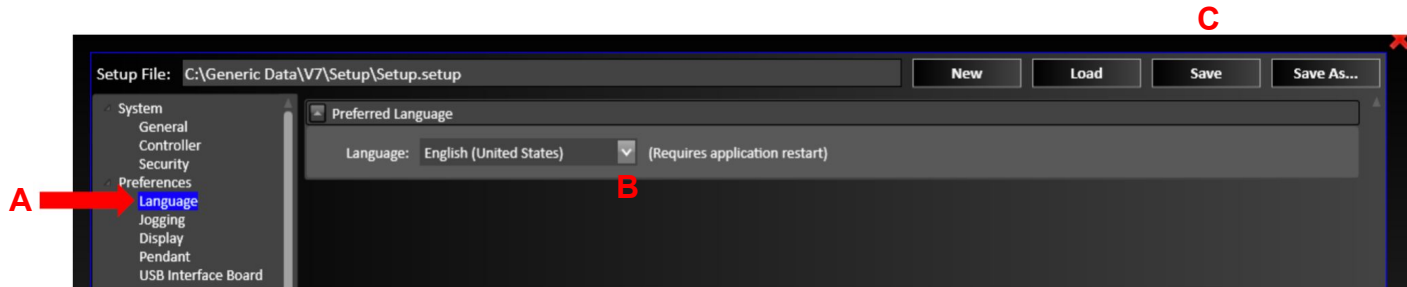
- English
- Spanish
- Italian
- French
- Portuguese
- German
- Japanese
- Dutch

Contact Information

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