

# FastCAM® Software

## Tutorial

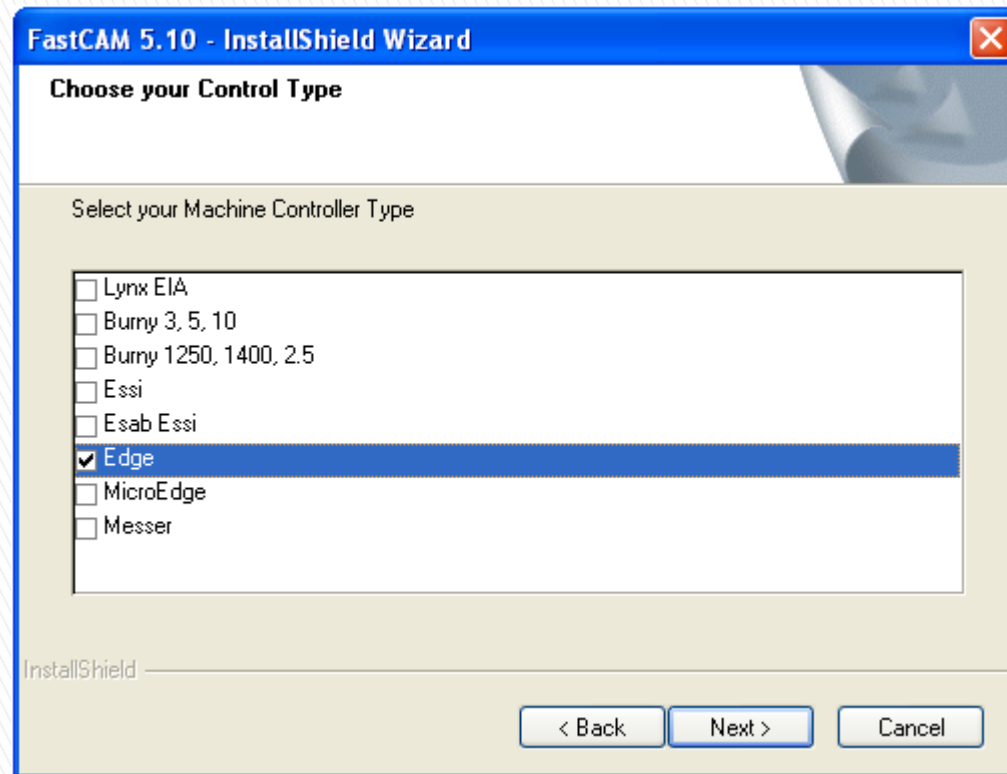
This tutorial supplements our classroom training material however due to the demand from remote customers unable to attend face-to-face training, we have made it freely available.

FastCAM Inc.  
FastCAM China  
FastCAM Pty. Ltd.

# Installation & Setup

## I. FastCAM Installation

Selection of the correct NC controller is an important step in the software installation. If uncertain, the correct controller can be verified on the cutting machine or by direct contact with the machine manufacture.



## Installation Notes

- During software installation, it is essential to select the correct controller and code format for your specific controller type. If you don't do this correctly you will get error problems. Machines and controllers are not always the same. You need to check the cutting machine documentation or ask the machine builder if you are not sure.

### Remarks:

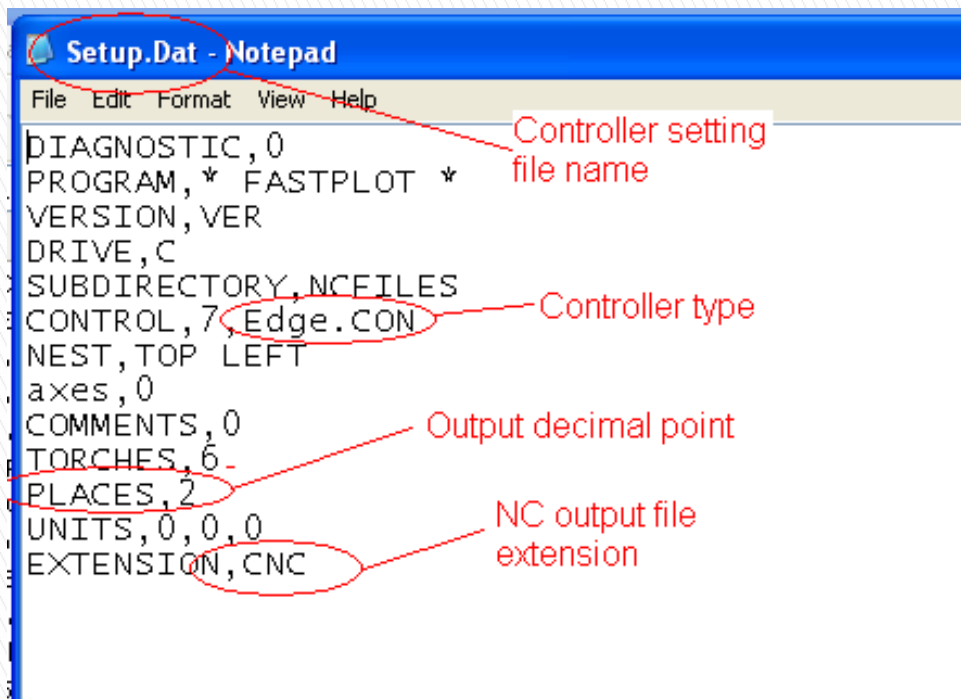
1. You may need to reinstall FastCAM in the future. It is important to make a backup copy of the controller setup file.
2. Other types of controllers for Oxy Flame and Plasma cutting system are EDGE, BURNY, FastCNC, START, KANGPU ... etc.
3. Common controller types for LASER cutting system are FAGOR, PA8000, SIEMENS ... etc.

# Installation and Setup

II. FastCAM System and Controller Settings are in the SETUP.DAT file and Control file these are normally located C:\program files\fastcam\  
Occasionally these files may need to be edited for specific machines. **If in Doubt** contact your supplier or FastCAM Service.

Note: NC output files do not always end in [filename].CNC

Other popular extension names are TXT and NC and some have no extension.



The image shows a screenshot of a Notepad window titled "Setup.Dat - Notepad". The window contains a text file with various settings. Several parts of the text are circled in red, and red lines point from these circles to text annotations on the right side of the window. The annotations are: "Controller setting file name" pointing to "FASTPLOT", "Controller type" pointing to "Edge.CON", "Output decimal point" pointing to "6", and "NC output file extension" pointing to "CNC".

```
DIAGNOSTIC,0
PROGRAM,* FASTPLOT *
VERSION,VER
DRIVE,C
SUBDIRECTORY,NCFILES
CONTROL,7,Edge.CON
NEST, TOP LEFT
axes,0
COMMENTS,0
TORCHES,6-
PLACES,2
UNITS,0,0,0
EXTENSION,CNC
```

# Installation & Setup Up

## III. Controller Setting (\*.con file, \* stands for the controller type, e.g. Edge.con)

```
Edge.con - Notepad
File Edit Format View Help

*
5. SYMBOL
/OPERATIONS/
* OPER          CONT  KERF  COL  LTYPE/  PLOTTER ON,OFF,START,STOP
*              CHAR
CUTTING,        2,   -1,  10,   1,       1,   M04,M03,
*MARKER,         2,    0,  13,   3,       1,   M9,M10,M11,M12
MARKER,          1,    0,  13,   3,       1,   M11,M12,,M00
TEXT,            6,    0,  14,   1,       1,   M11,M12,,M00

/KERF RIGHT/
G42
/KERF LEFT/
G41
/KERF OFF/
G40
Setting codes for
kerf compensation

* GENERAL UTILITY STATEMENTS
/OPENING STATEMENTS/
%
/COMMENT ON/
/COMMENT OFF/
/RAPID ON/
/RAPID OFF/
/CLOSING STATEMENTS/
M2
/HALT/
/START OF NEST/
/END OF NEST/
M2
program end code

/DXF/
1,1
2,2
relative layer setting
```

## Notes:

Editing the controller setup file.

SETUP.DAT is an important file for setup of the controller which stores the name and type of the controller together with the related settings.

Changing the content of this file may cause operational problems. Details of this file can be found in the FastCAM reference manual.

Note: For accuracy, precision LASER cutting is set to 0.01 mm in FastCAM.

## Notes- Editing the controller setup file \*.CON

\*.CON (e.g. FAGOR.CON, EDGE.CON) is another important file for setup of the controller because it stores commands for the cutting machine. For further details, please check the FastCAM reference manual or contact our technical staff. Content of the file should not be changed if the NC Codes and their exact meaning are not understood.

### Important:

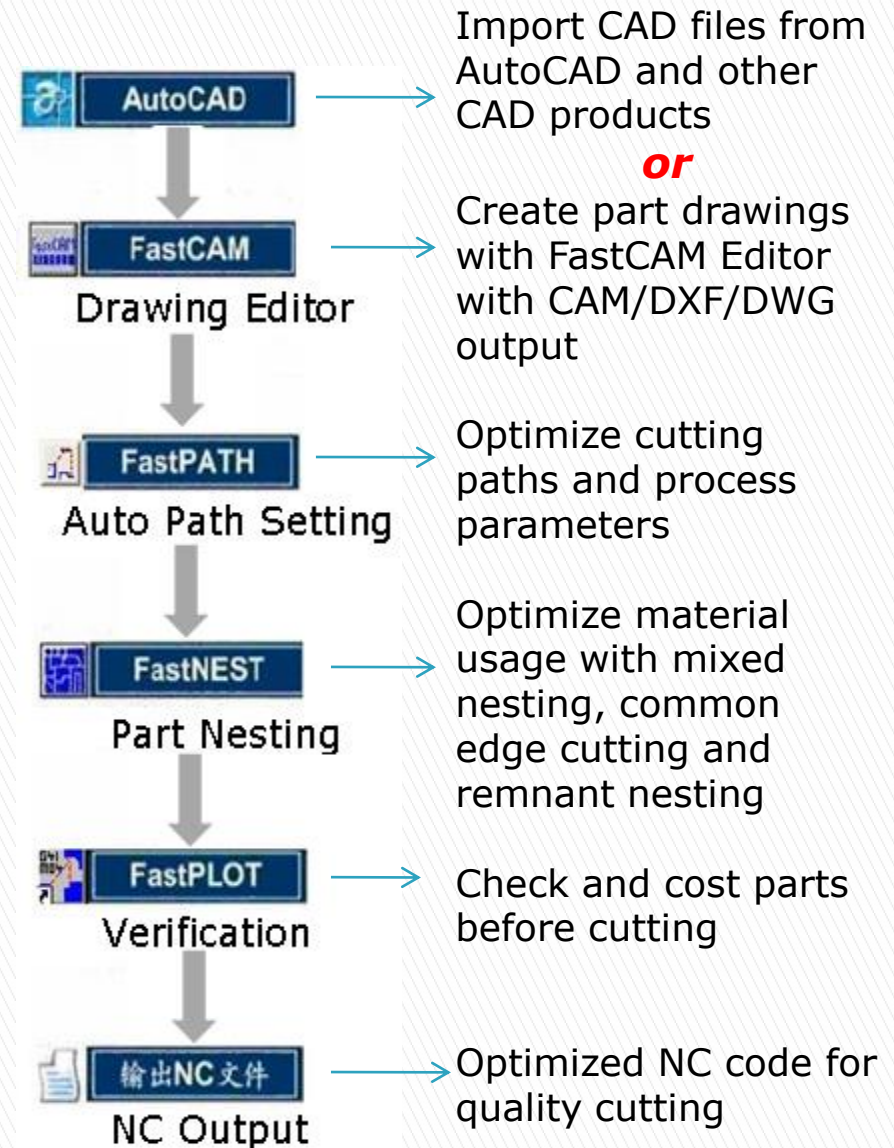
For software re-installation in the future, please make a copy of the 2 controller files (SETUP.DAT and \*.CON) and keep in a safe place. These documents can be found in the FastCAM installation directory eg: C:\Program Files\FastCAM

# FastCAM System Overview

## I. FastCAM Application Process

Draw -> Nest -> Cut  
All in One program.

We discuss, illustrate and demonstrate each function.





# Content

## **Module 1 The Part Drawing - 45 minutes**

- Lesson 1: FastCAM Drawing Editor Quick Study (10 minutes)
- Lesson 2: FastCAM Drawing Editor Full Study (30 minutes)
- Lesson 3: Working with CAD files (30 minutes)

## **Module 2 NC Path Setting - 45 minutes**

- Lesson 3: Manual Path Setting (15 minutes)
- Lesson 4: FastPATH Automatic Path Setting (30 minutes)

## **Module 3 Part Nesting - 45 minutes**

- Lesson 5: Manual Part Nesting (15 minutes)
- Lesson 6: Automatic Mixed Part Nesting (30 minutes)

## **Module 4 Optimized Cutting– 45 minutes**

- Lesson 7: Automatic Common Cutting (15 minutes)
- Lesson 8: Bridges for Plasma Cutting (15 minutes)
- Lesson 9: Automatic Continuous Cutting (15 minutes)
- Lesson 10: Nesting exercises

## **Module 5 FastCAM Solutions**

## **Appendix: Technical Notes**

**- END -**

## FastCAM Drawing Editor - Quick Study



## FASTCAM Drawing Editor - Full (self) Study

### Fundamentals:

- ▶ Point, Line and Arc
- ▶ Erase, Trim, Blocks, View, Utility and Text
- ▶ Text, Verify and Standard Parts

## Drawing

The FastCAM drawing editor is a simple 2D drawing tool which requires little or no previous CAD experience.

FastCAM has drawing functions similar to AutoCAD without the complexity. It has been developed specifically for steel cutting and for use by trades people.

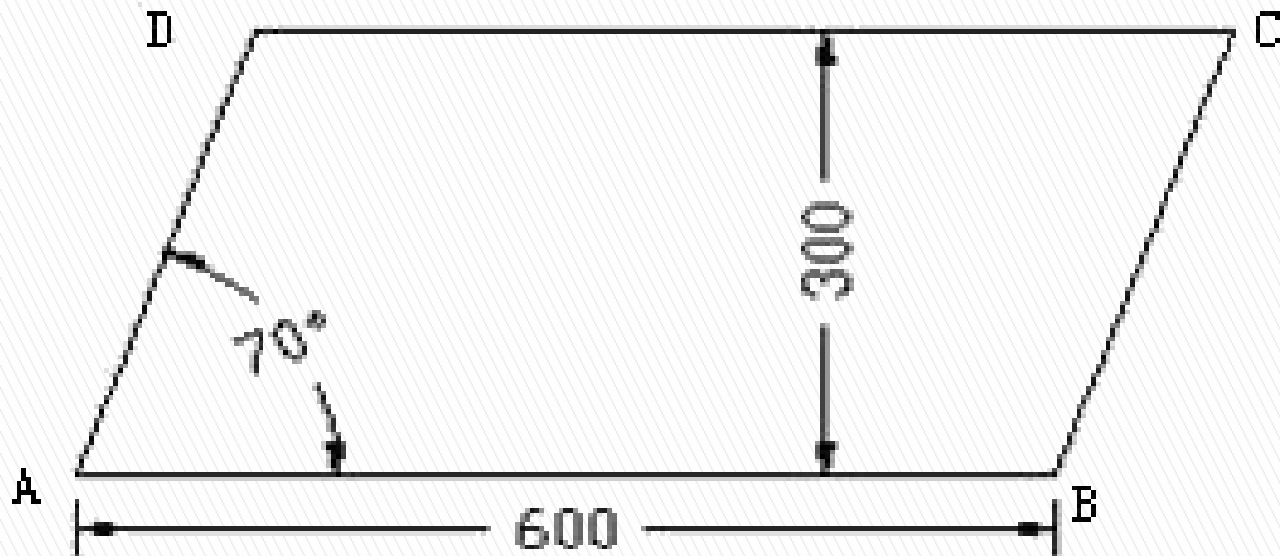
It also provides many standard features required commonly by industrial cutting such as different types of bridges and stitches. With LASER cutting, there is choice for process techniques such as internal corner to fillet, external corner with fillet or chamfer, contour to slot and hole to point.

### CAD Note:

1. For people already familiar with a CAD program, it is not necessary to learn the FastCAM drawing editor. However to prevent cutting problems, it is important to use the inbuilt data optimizer and to verify paths are closed before nesting. Please browse through this section or go directly to Lesson 2 (Slide 31).

## 1.1: Quick Study A: Draw a Basic Shape

### 1. Drawing exercise

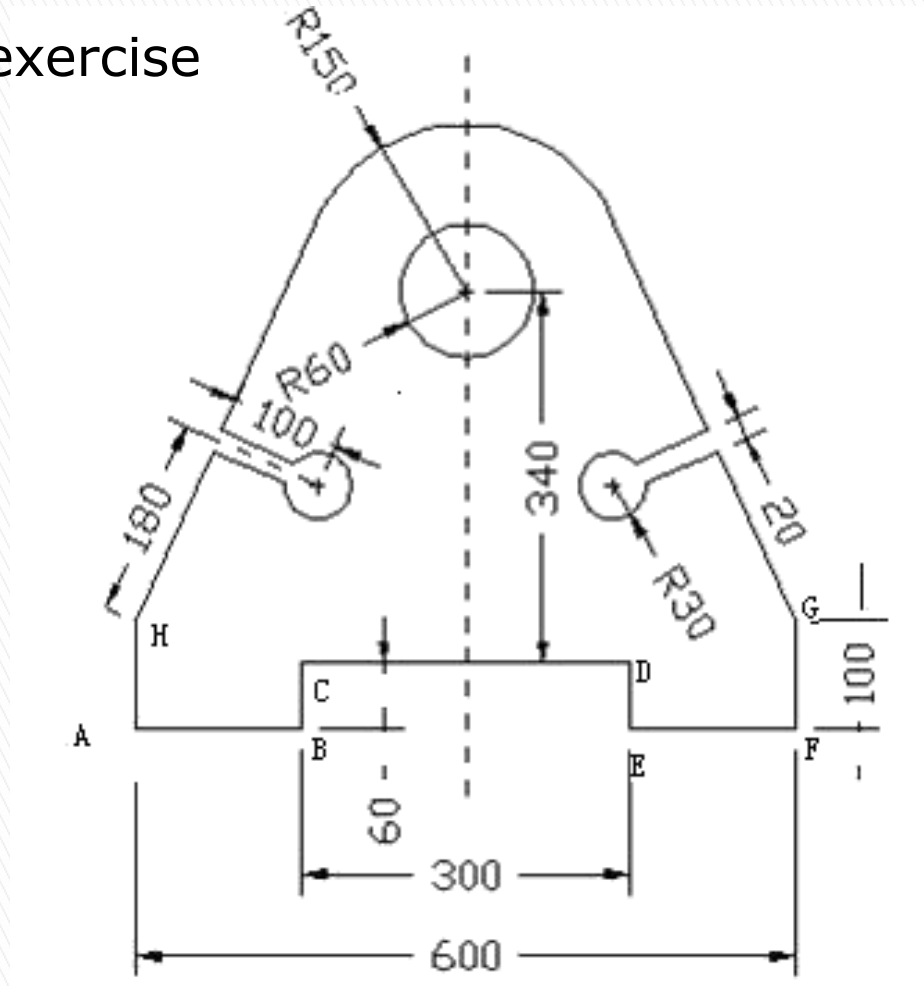


## Instructions to Create a Basic Shape:

1. Select 'Line - 2 points' button under the menu bar, right click on any blank screen area and select 'Screen position'.
2. Left click on screen to define the first point 'A' followed by right click to select 'incremental co-ords'.
3. Using 'A' as the reference point to define point 'B', enter (600, 0), enter and exit to main menu.
4. Select 'Angle only' from the 'Line' menu, left click 'A' as the reference point and select 'relative to' from pop up.
5. Left click on previous drawn line close to 'B', enter 70 for degree in the dialog box and press 'ESC' until back to main menu.
6. Select 'Parallel at dist.' from the 'Line' menu, left click on line AB and enter 300, click on screen above line AB to indicate the required side and select infinite line from pop up.
7. Using 'Angle only' from the 'Line' menu, pick 'B' as the reference point to draw the line BC.
8. Select 'Smart trim' from the 'Trim' menu and left click on lines outside of the required shape to complete this exercise.

## 1.1: Quick Study B. – Draw a 'Widget'

## 1. Drawing exercise

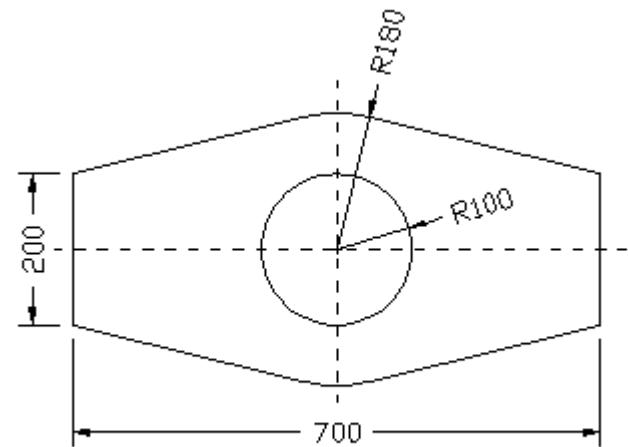
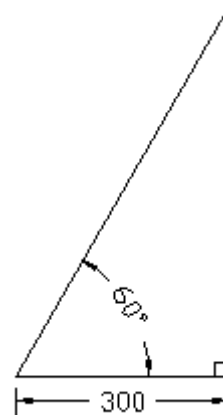
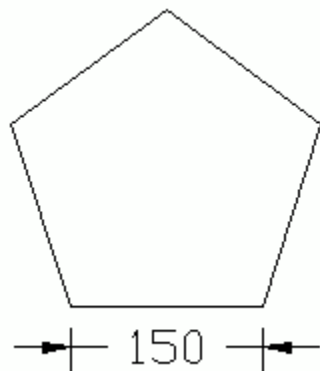
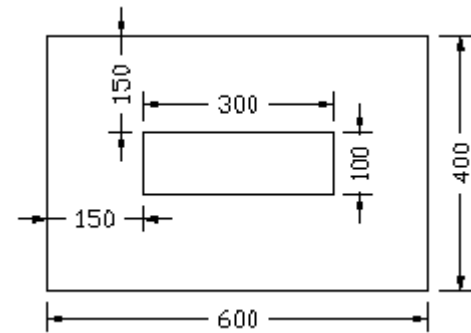
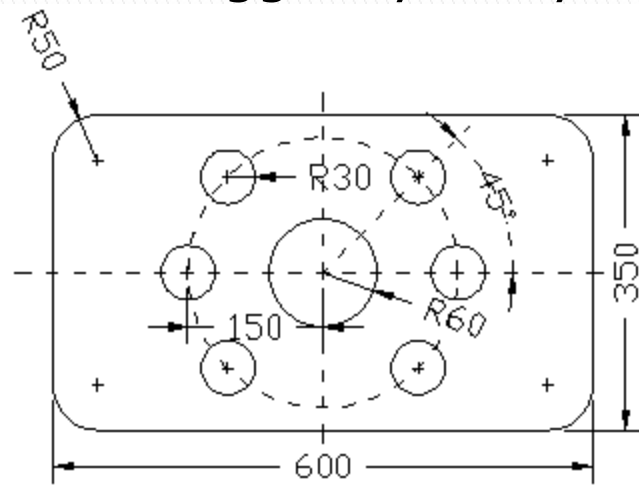


## Instructions to create a 'Widget' (JACKSON PLEASE BREAK DOWN INTO STEPS)

1. Select 'Line 2 points' button from menu, click on screen to define point 'A' followed by right click and select 'incremental co-ords' from pop up. Enter 150 and 0 in dialog box to complete drawing of line AB. Click on 'B' as the reference point, right click to select 'incremental co-ords' and enter 0, 60 for line BC. Click on 'C' as the reference point, right click to select 'incremental co-ords' and enter 300, 0 for line CD. Click on 'D' as the reference point, right click to select 'incremental co-ords' and enter 0, -60 for line DE. Click on 'E' as the reference point, right click to select 'incremental co-ords' and enter 150, 0 for line EF. Click on 'F' as the reference point, right click to select 'incremental co-ords' and enter 0, 100 for line FG. Select 'Line 2 points' button from menu, click on 'A' as the reference point, right click to select 'incremental co-ords' and enter 0, 100 for line AH. Select Ring from the Arc menu, enter 120 and 300 as diameter of rings in the dialog box. Right click to select point A as the reference point and enter (300,400) as coordinate of ring centre. Select 'Tan to Circle' from the Line menu, click on outer circle of the ring, select given point and click on point H followed by right click to escape. Complete drawing of the tangent to point G with the same principle. Select Full Circle from the Arc menu, enter 180 as radius and pick point H as centre with an auxiliary circle drawn intersecting at I. Select 'At Rt Angles' from the Line menu and click on I with a perpendicular line drawn. Draw another circle with radius 100 and centre at I. Locate centre of circle with radius 30 from the intersection and draw the R30 circle. Create 'parallel lines at distance' at point I from the Line menu. Continue to finish rest of the geometry and use smart trim to complete this exercise.

## 1.1: Other Fast Study Examples:

Other we suggest you try:

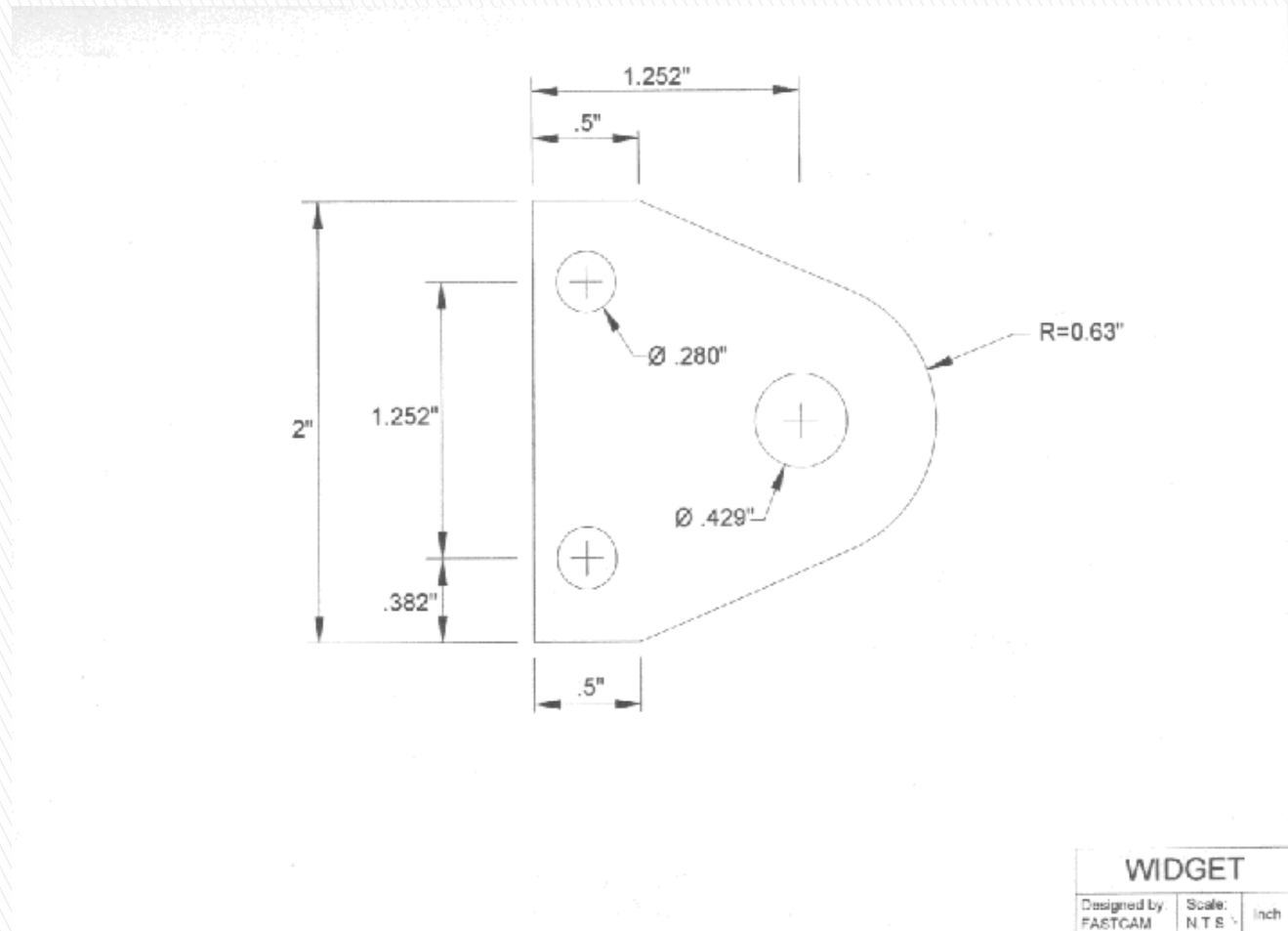




## 1.2. FastCAM Drawing Editor – Full (Self) Study

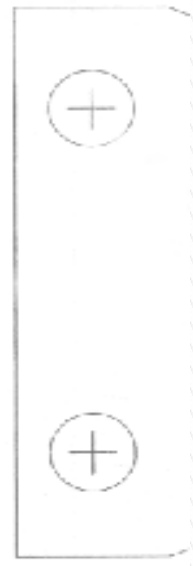
### WIDGET

Here you will go through the process of drawing a part in FastCAM® based upon a scaled part drawing. For example you might receive a drawing from a customer like the one below:



With any drawing you do in FastCAM®, the first step is to identify the individual shapes within your part. This may take some time at first, but with practice and experience, you will learn how to quickly evaluate and determine what entities are necessary to create a part.

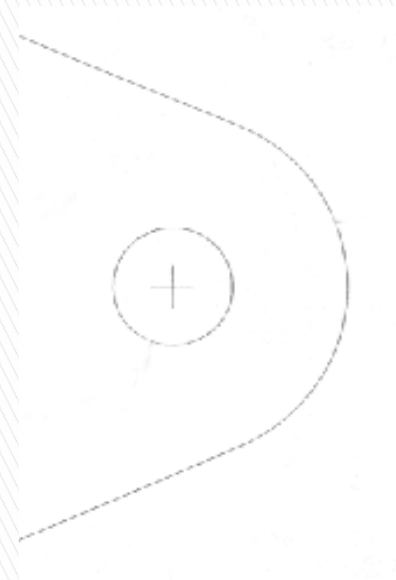
In our WIDGET, it is important to notice there is a rectangle on the left side within the part. This rectangle also has two smaller internal circles. On the right side, you can see a 2-circle ring. The rectangle and ring appear to be blended together with two lines.



*Rectangle*

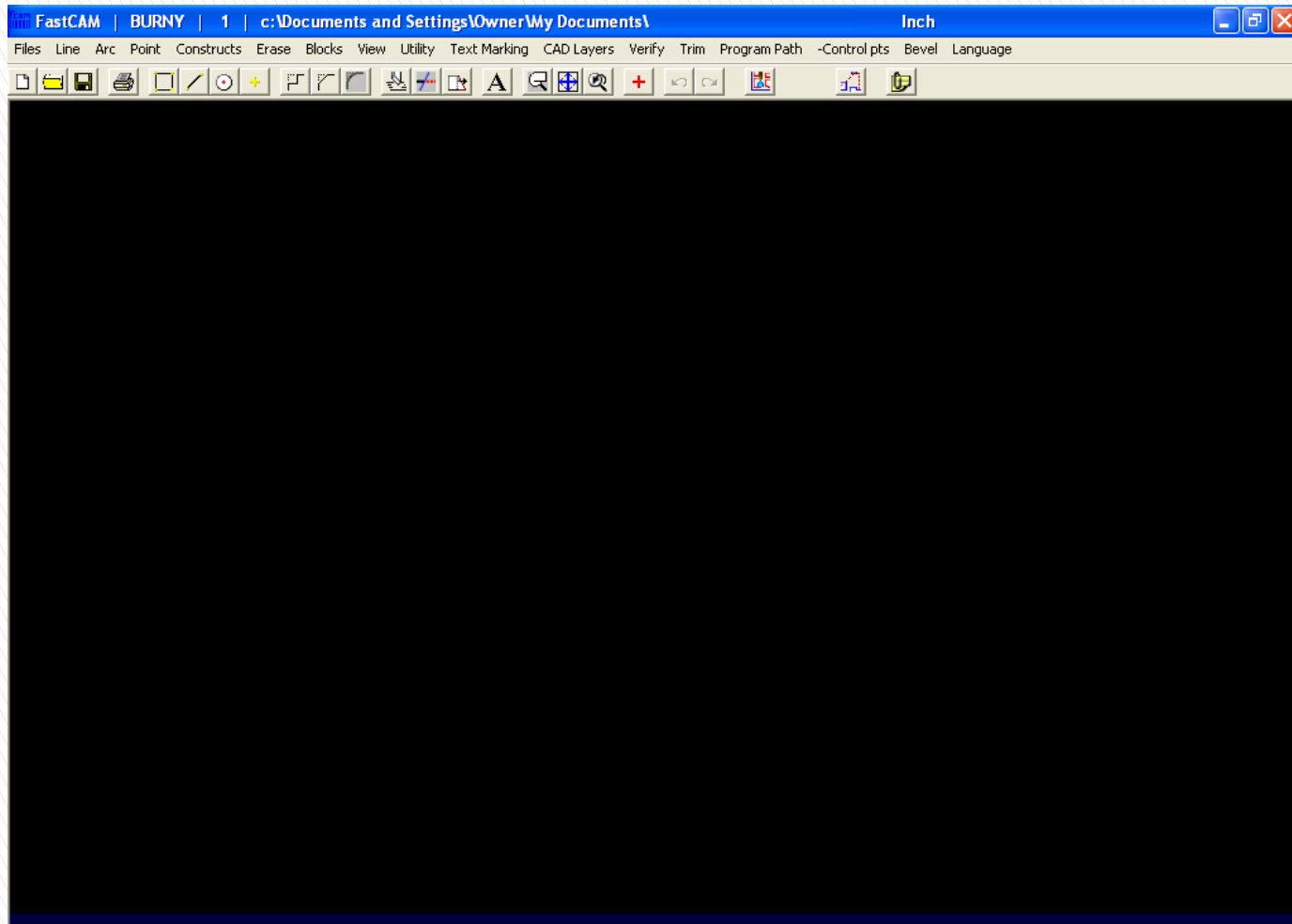


*Connecting Tangent Lines*

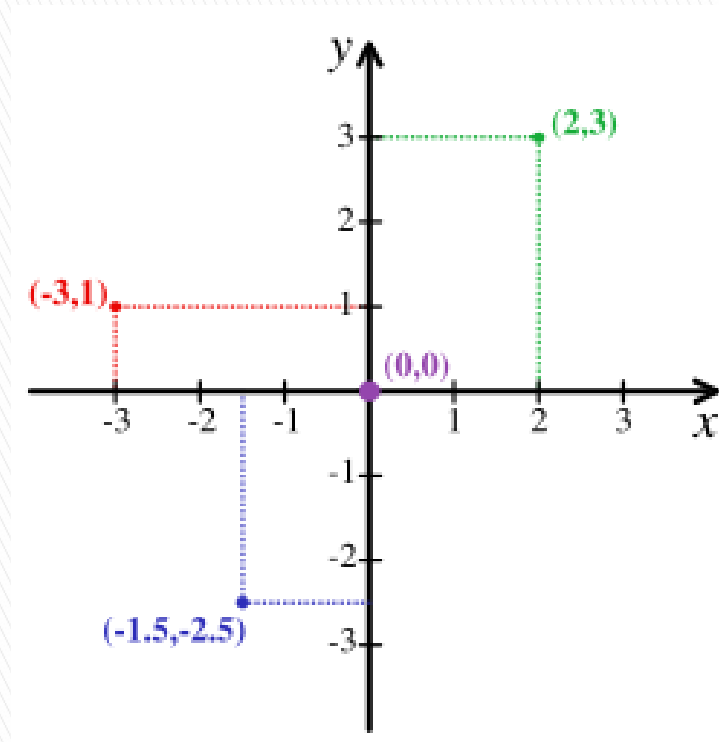


*Ring*

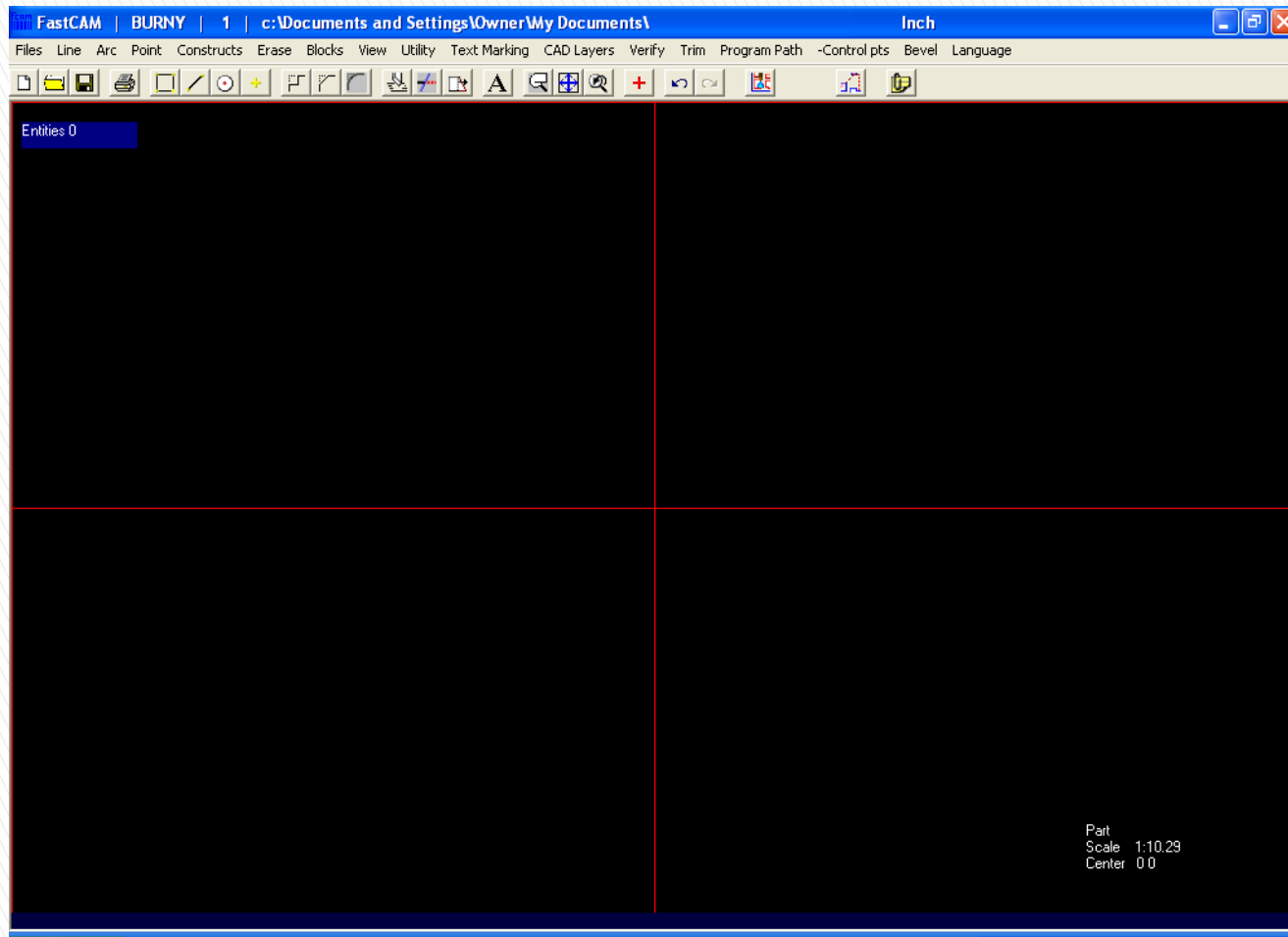
Once you have identified the different entities in the part, we start by running the FastCAM® drawing editor. Upon opening FastCAM®, you should see the screen below.



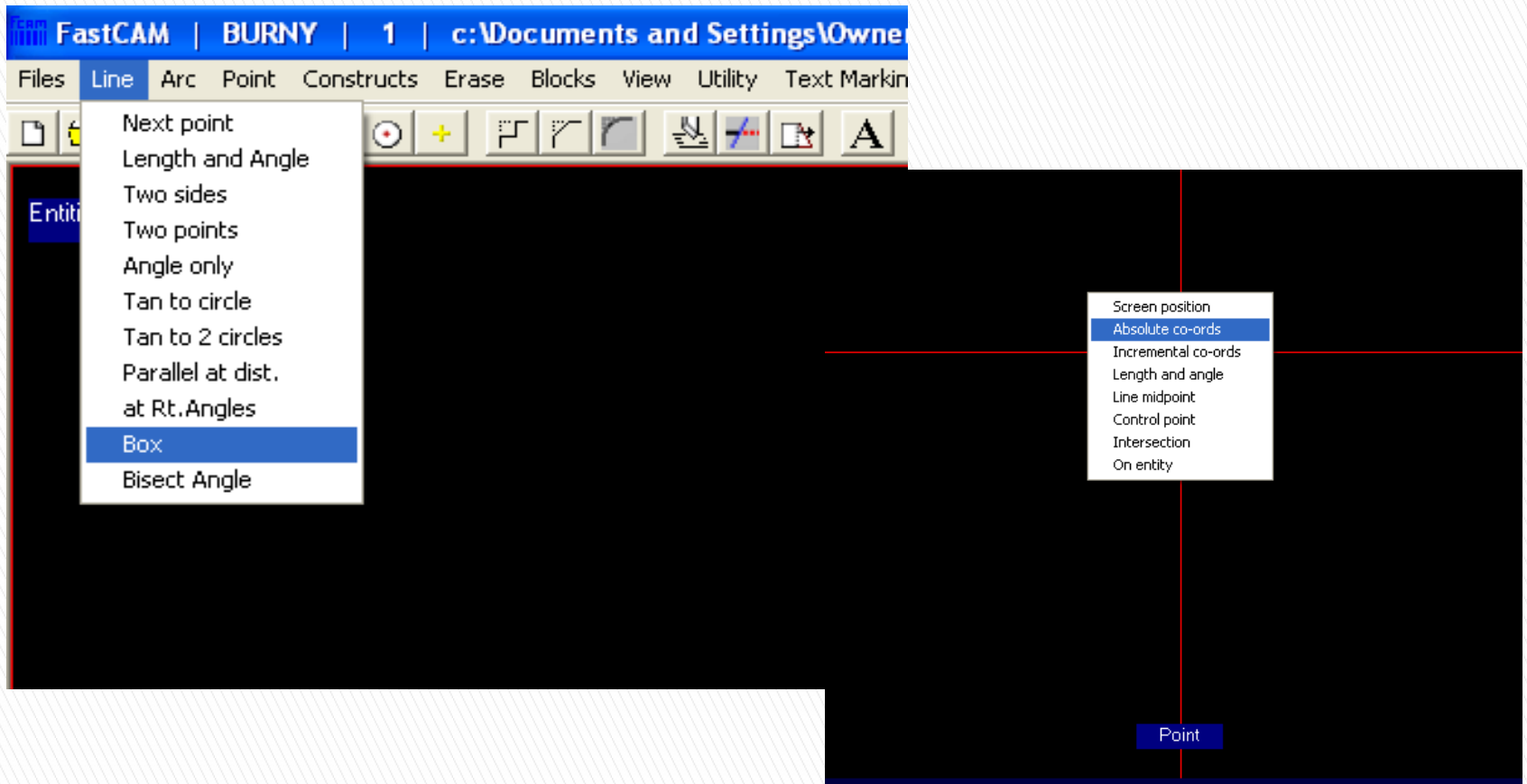
To create drawings to scale, it is easiest to use absolute coordinates. Absolute coordinates are based upon the Cartesian coordinate system as shown below.



If you are unfamiliar with the Cartesian coordinate system, it is a good idea to review or practice using this principle. In FastCAM®, you can choose to display the x and y axis to assist you in drawing your parts. Click on the View menu and select Change Display. From here, check the box for Axes and then press ENTER.

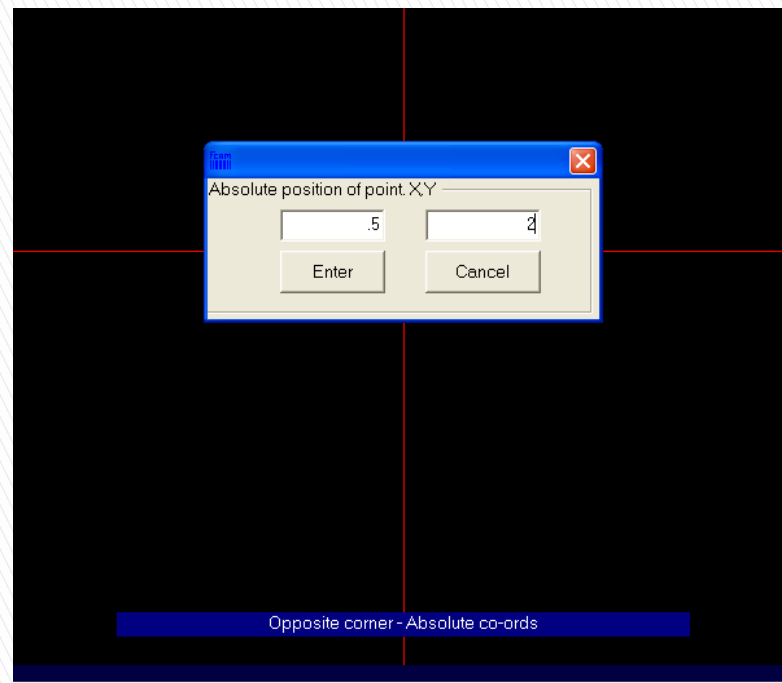
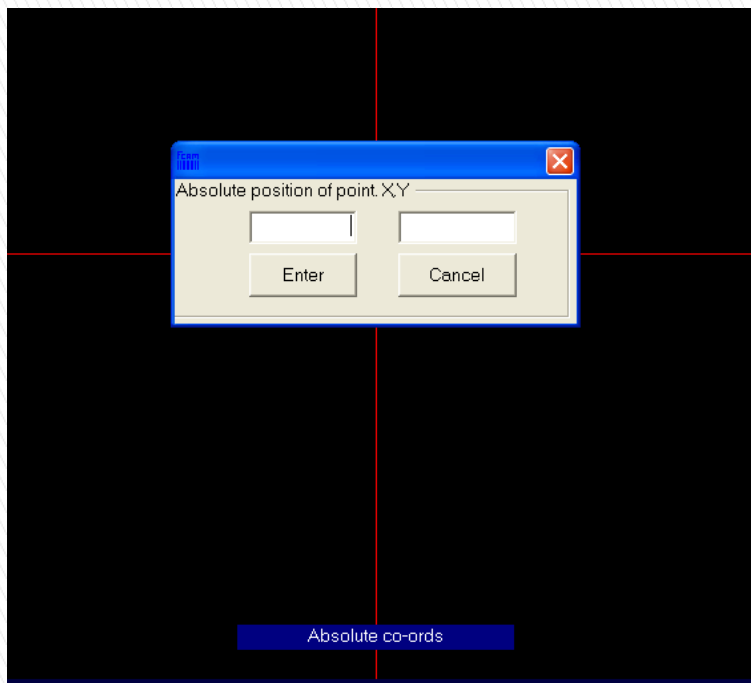


You can start by drawing the rectangle. It is best to do this using the Box feature. From the Line menu select Box.

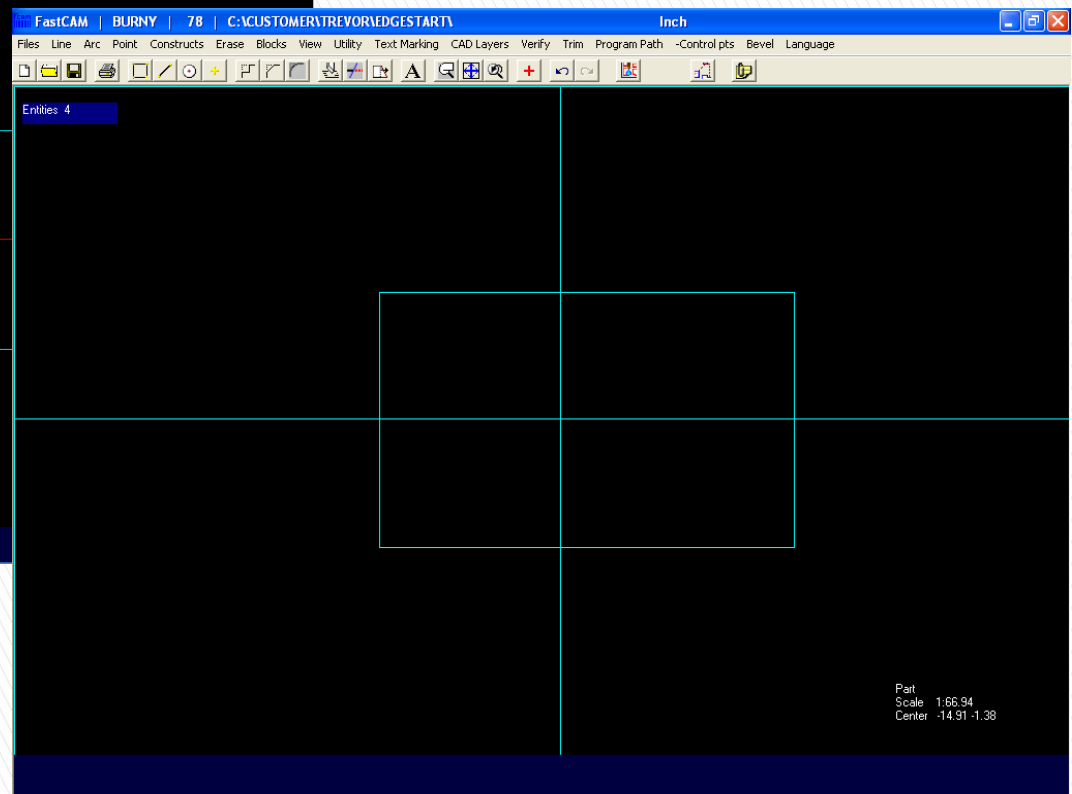
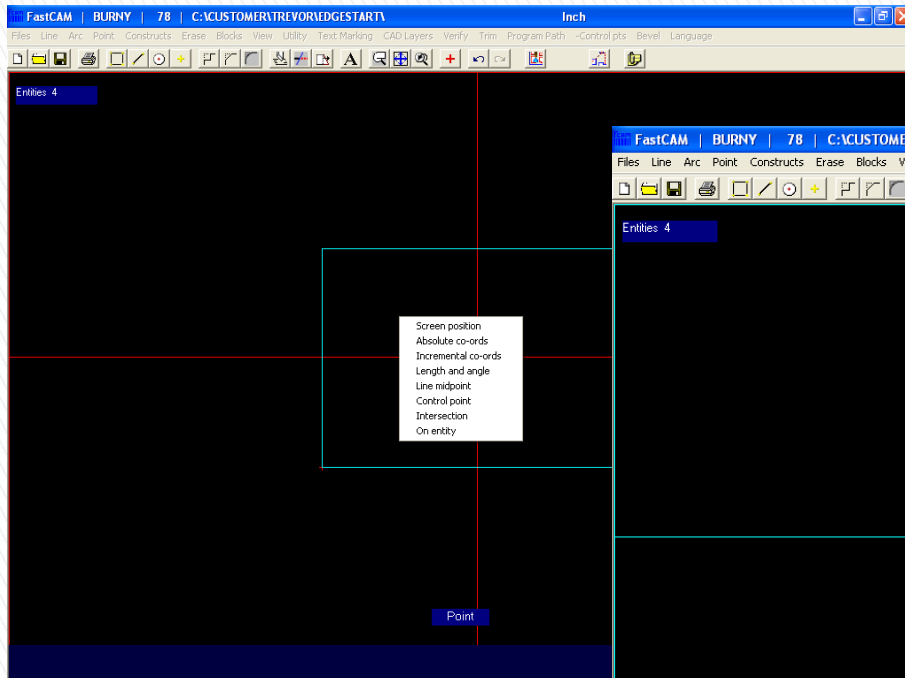


By default, FastCAM® is set to draw a box using screen positions. This function allows you to quickly draw rectangles by simply clicking on the screen but does not allow for parts to be drawn to scale. To draw your part to scale, you must switch to the absolute coordinates by right clicking once on your drawing space and selecting Absolute Coordinates.

You can now enter the x,y coordinates for two opposite corners of your rectangle. Based upon your drawing, you can enter two diagonally opposite corners of your box. You can use a first point of (0,0), a second point of (0.5,2) and click 'Enter'.

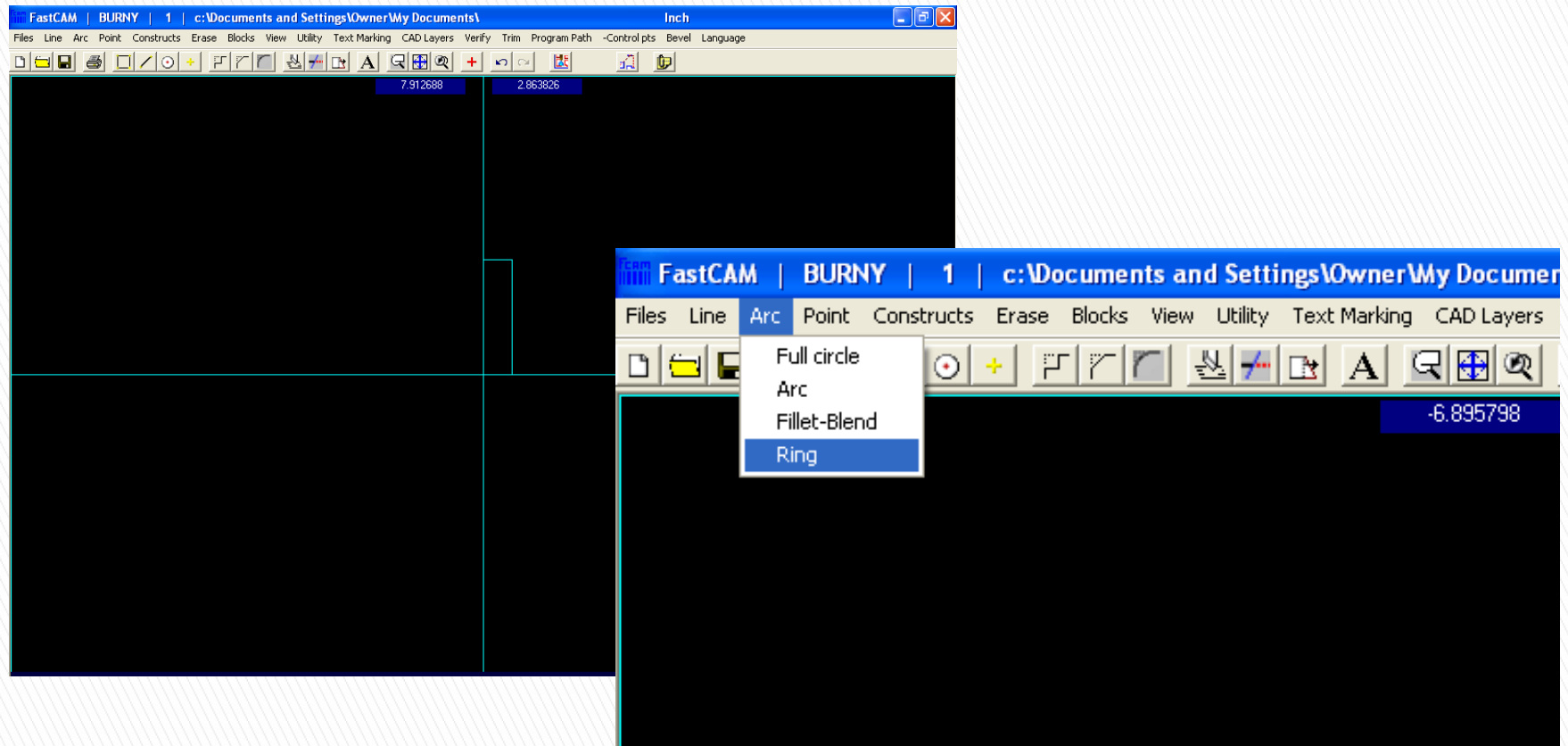


*Many of the drawing functions in FastCAM® are repetitive. To get out of any command or popup window, you will need to right-click until the menu bar is back to active.*



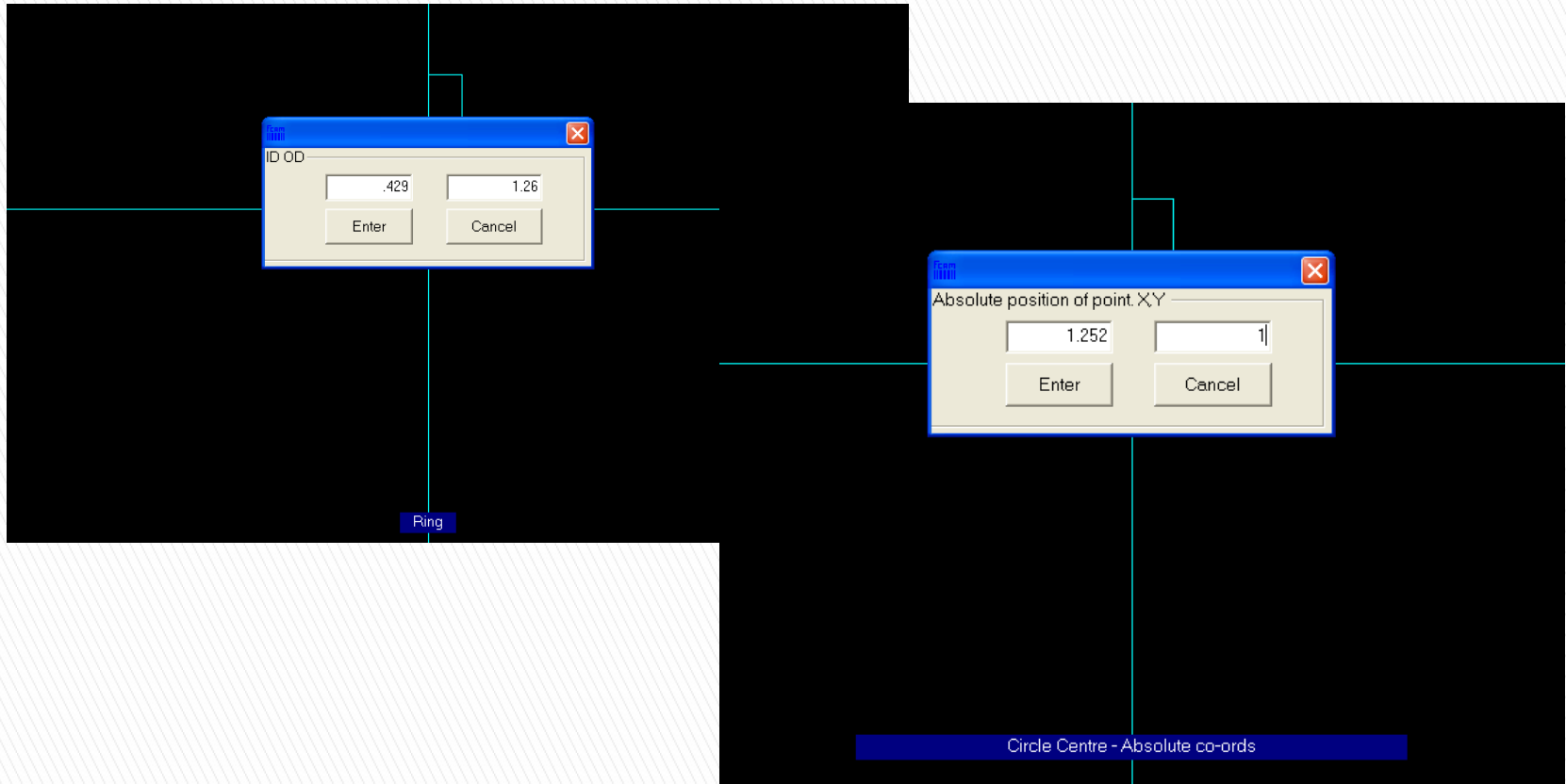


The rectangle should now be drawn to scale and appear on the screen as shown below. When drawing items, FastCAM® often automatically prompts for a second item to be drawn. If this is not desired, you may simply hit cancel and right click on the screen repeatedly to get out of any popup windows.



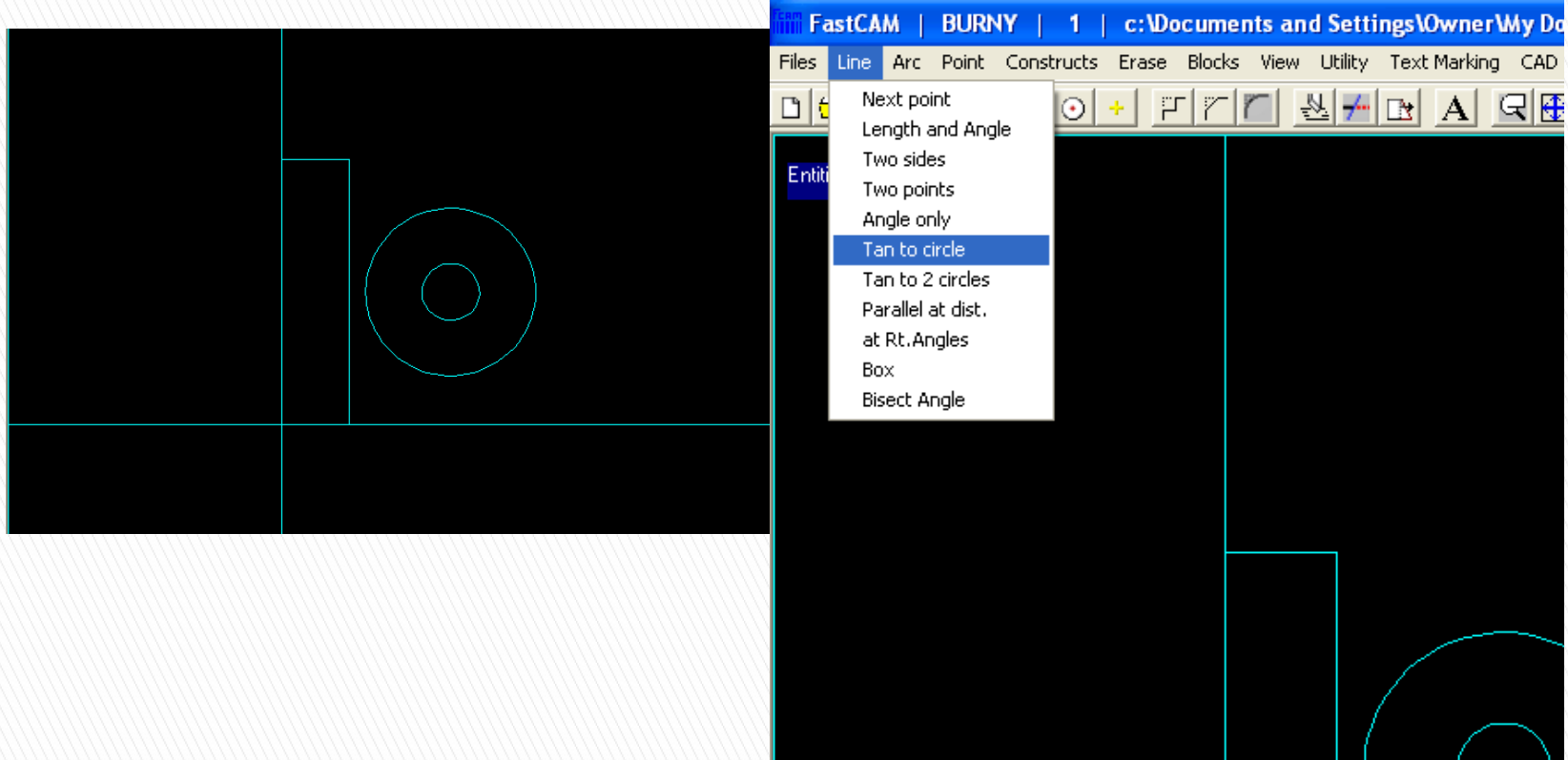
The next step would be to create the other side of our drawing (the rings.) You could draw two full circles with the same center, but it is easier to use the Ring feature under the Arc menu.

FastCAM® will prompt you for an inside diameter and an outside diameter of the two circles that make up the ring. Here you can key in .429 for the ID, 1.26 for the OD and click 'Enter'.



Next, you will be prompted to select the center point for your ring. Again, you can use absolute coordinates, key in (1.252,1) and click 'Enter'.

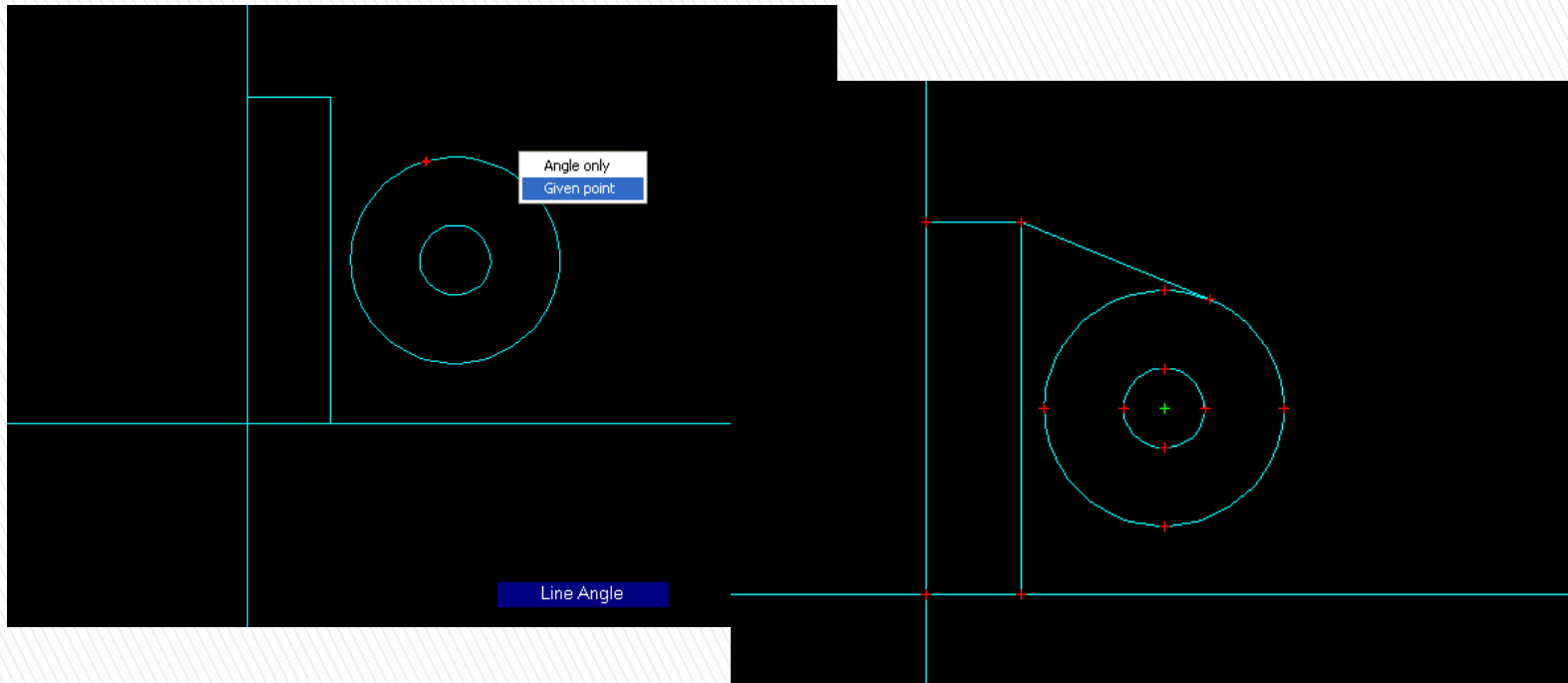
You should now see a full ring just to the right side of your previously drawn rectangle. If at anytime you need to refresh your screen (if the image of your drawing is not clear), hit the Enter key once.



Finally, you need to draw lines to connect the rectangle to the ring. There are many choices for drawing a line however, you can see from the scaled drawing that you need to use tangent lines. Therefore, you need to select the Tan to circle option under the Line menu.

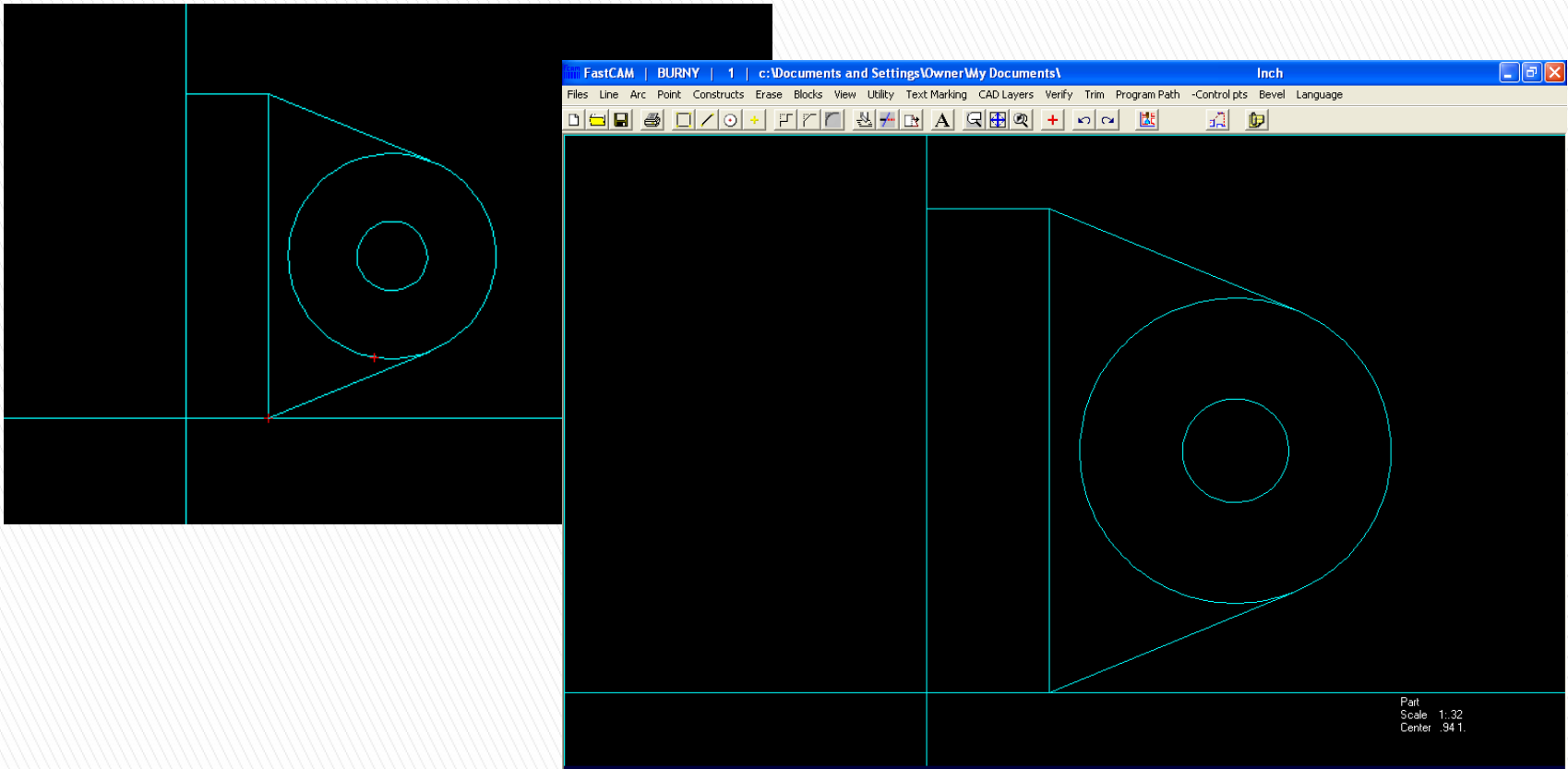
You will first need to select the arc (circle) in order to start the line. To do this you must click on the outer circle on the nearest side to where the line will intersect.

*Note : Horizontal infinite line will be created by left-click of mouse cursor at the approximate 6 and 12 O'clock positions. Likewise, vertical infinite line will be drawn by left-click of mouse cursor at the approximate 3 and 9 O'clock positions.*



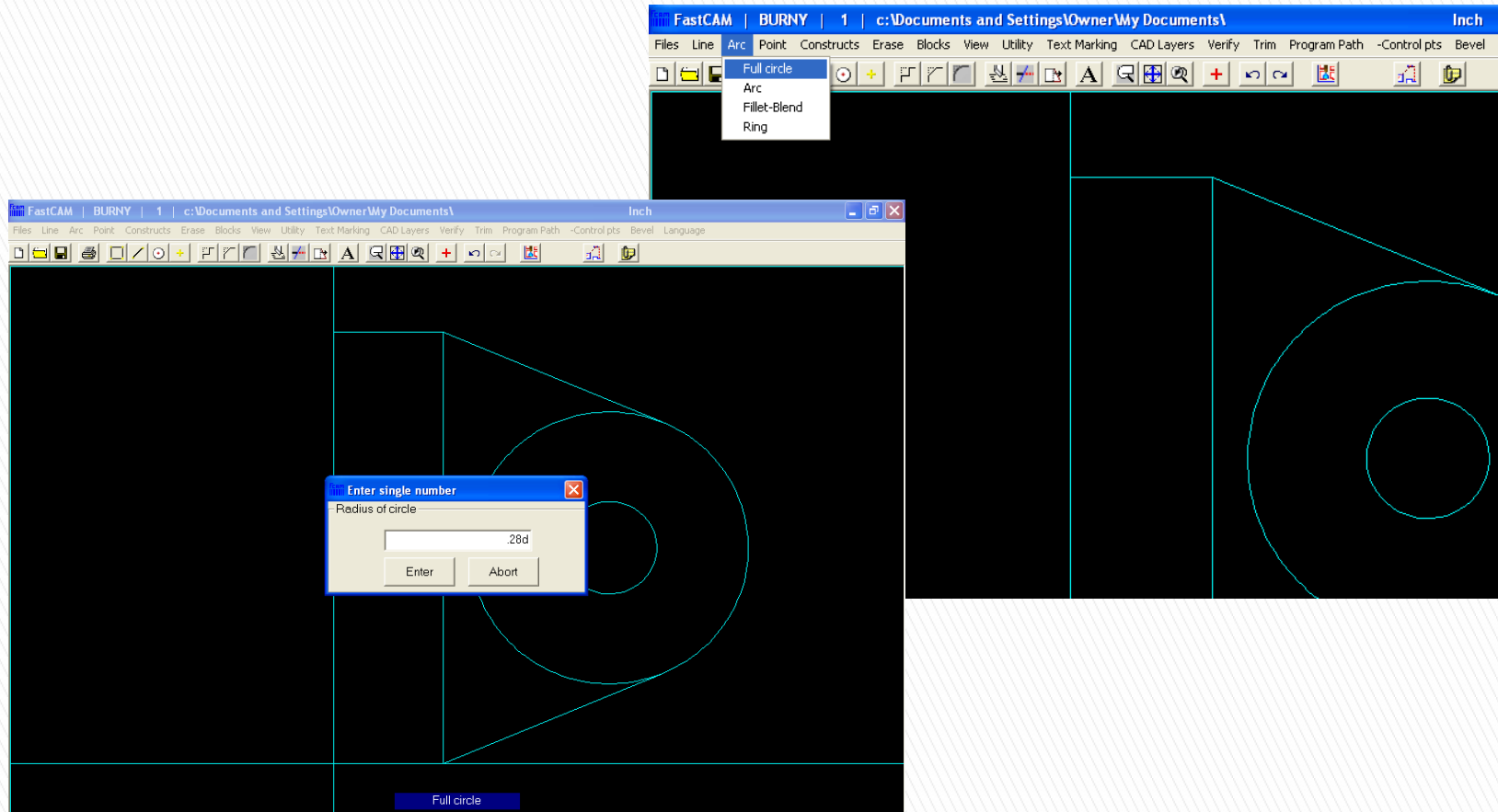
FastCAM® will then prompt you to choose an angle or a given point as an endpoint of your line. In this situation, you will want to select Given point and then click on the upper right-hand corner of the rectangle. After selecting Given point, the program will automatically switch to Control Points (from Absolute Coordinates) as can be seen by the prompt message on the screen. This means that as long as you click near the point, the line will “snap” exactly to this point. These points are called control points and you can see them in green and red below.

Repeat these steps for the other connecting line and you should then have the same as the drawing below. Be sure that you clicked at the appropriate point on your entity and that the correct tangent line appears.



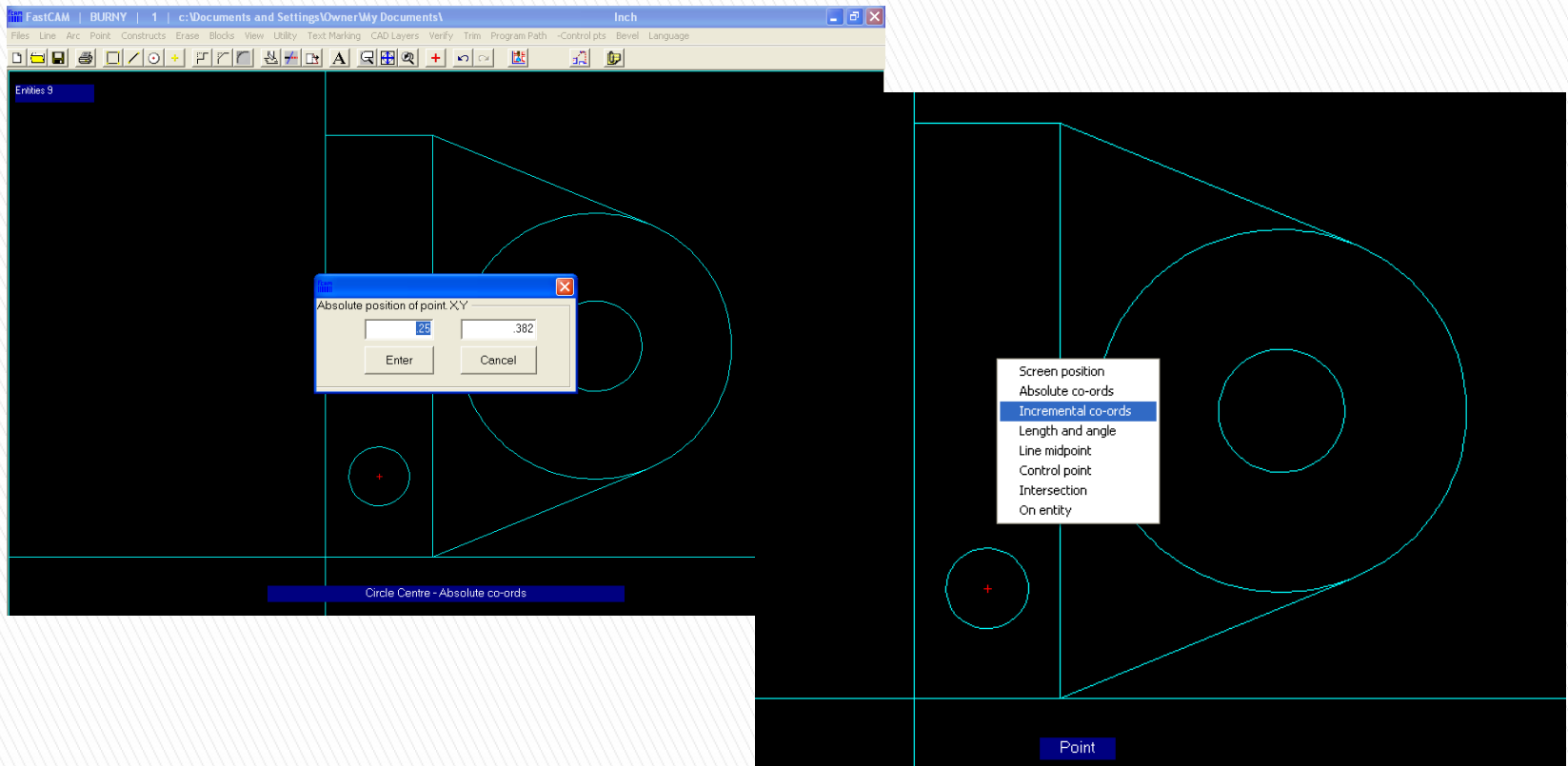
Our part is now drawn except for a couple of internal circles. First, you can zoom into this part by choosing Autoscale under the View menu, or by hitting the number 5 on the numerical keypad of your computer.

To draw a circle, go back up to the arc menu and select Full circle.



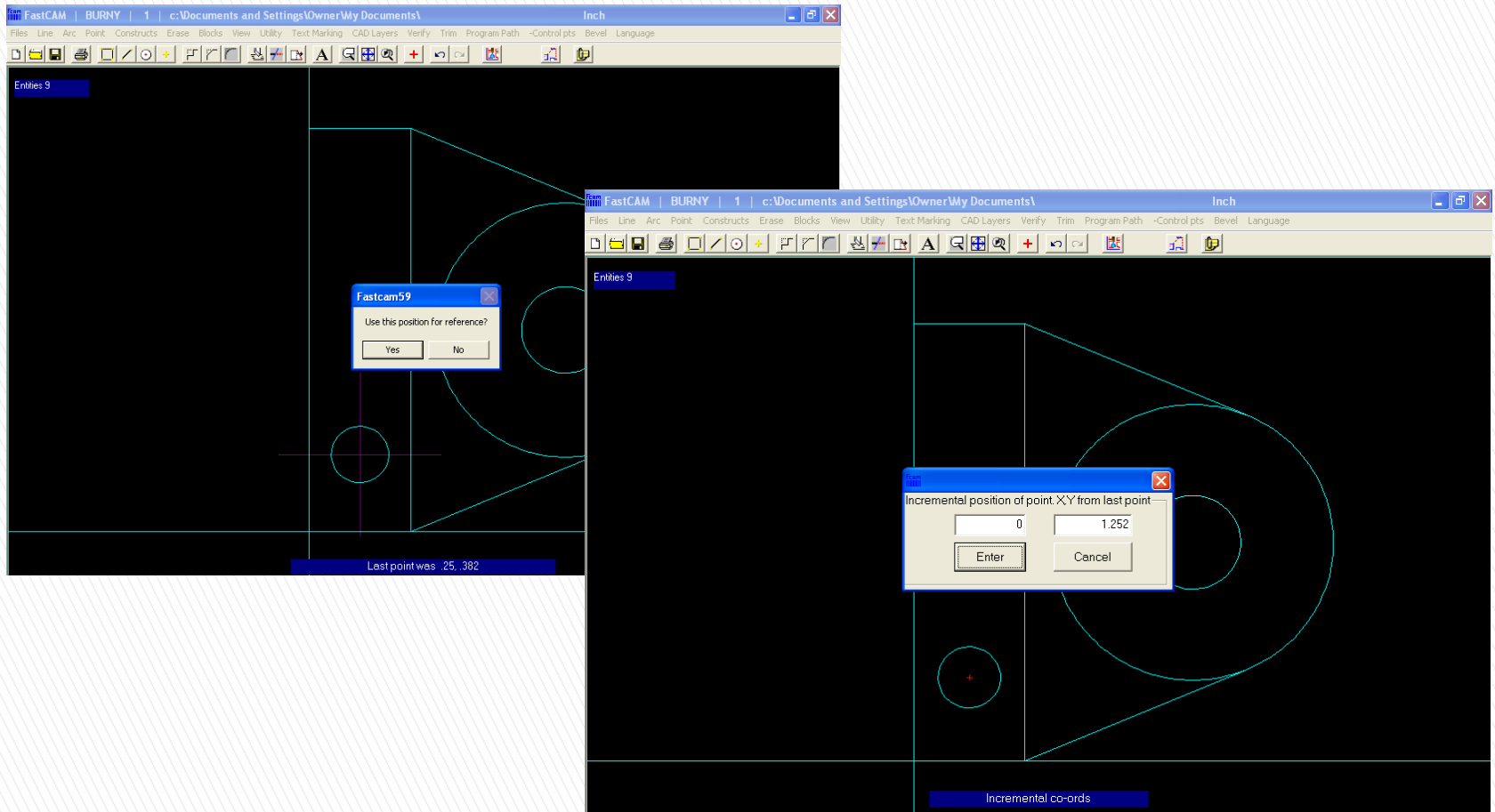
FastCAM® will ask for the radius of the circle. Since we have the diameter, we can simply enter the diameter followed by the letter "d" as shown below and click 'Enter'.

We can then enter the absolute coordinates of the lower internal circle based upon our scaled drawing. After you enter the diameter, you may have to right click and select absolute coordinates.



Many drawing functions in FastCAM® are repetitive, and the Circle function is one of them. Once we have drawn the first FastCAM® prompts us for another set of Absolute Coordinates for the next circle location. In this example it is easiest to use Incremental Coordinates, based on the drawing supplied. To do this, right click once and choose to use incremental coordinates.

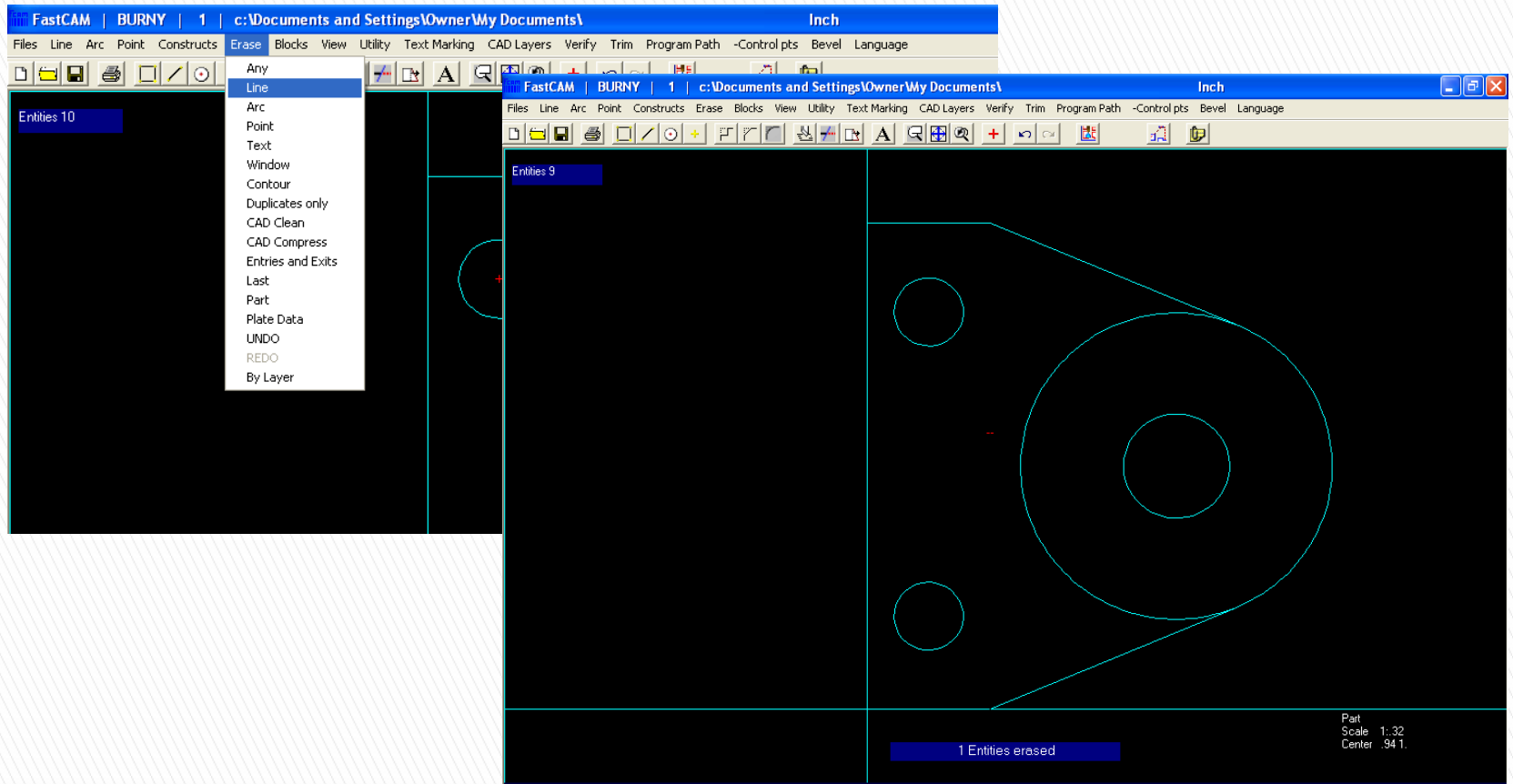
FastCAM® will ask if you want to use your previous point as your new reference point. Select yes, and enter coordinates based on that the point selected is now your reference of (0,0).



Now we need only the distance between the two circles, or the change in y value.

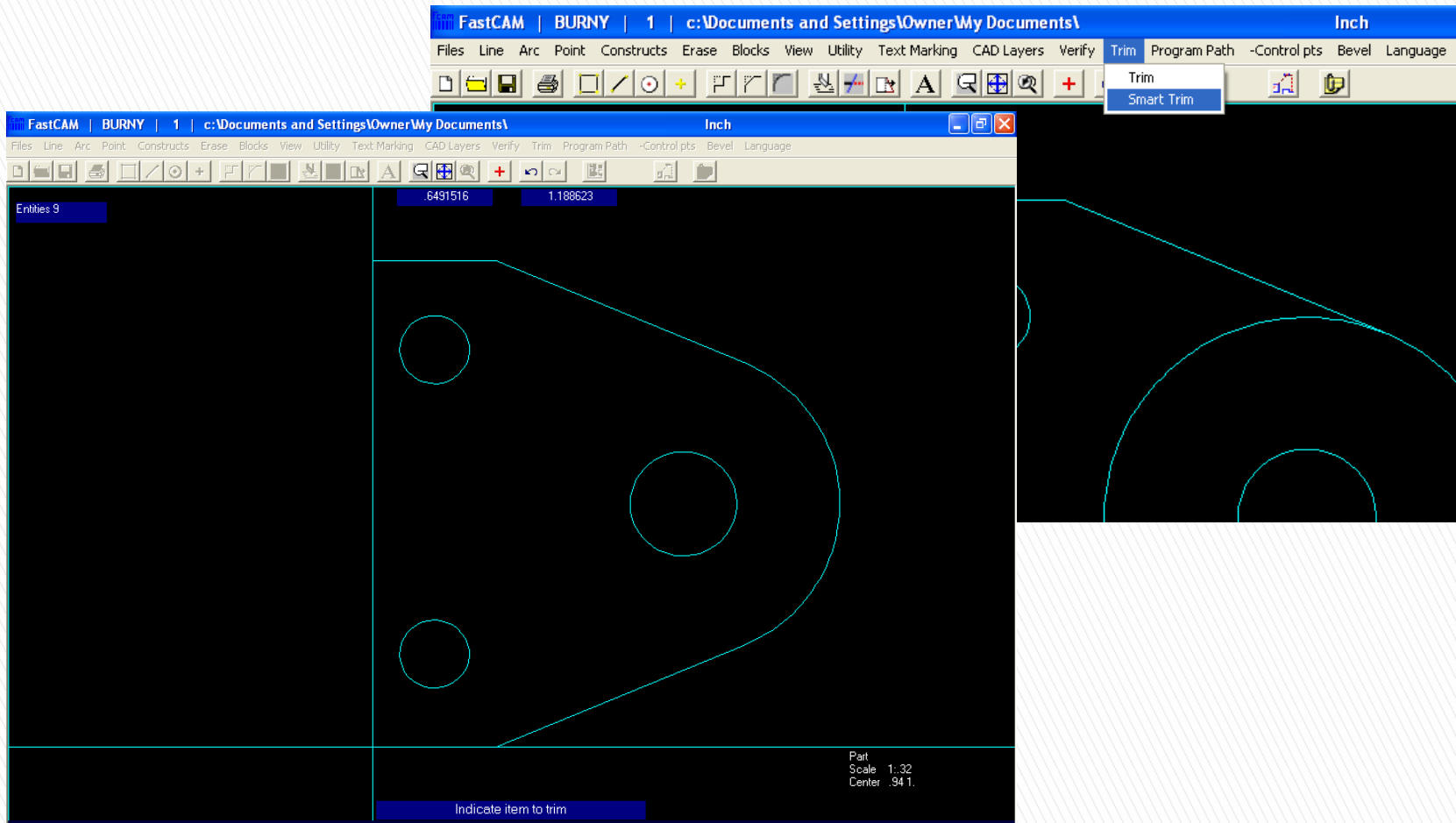


We now have a completed drawing, but need to erase some internal lines and arcs that are not in our final drawing. To erase a line which makes up part of the rectangle, go up to the Erase menu and choose Line.



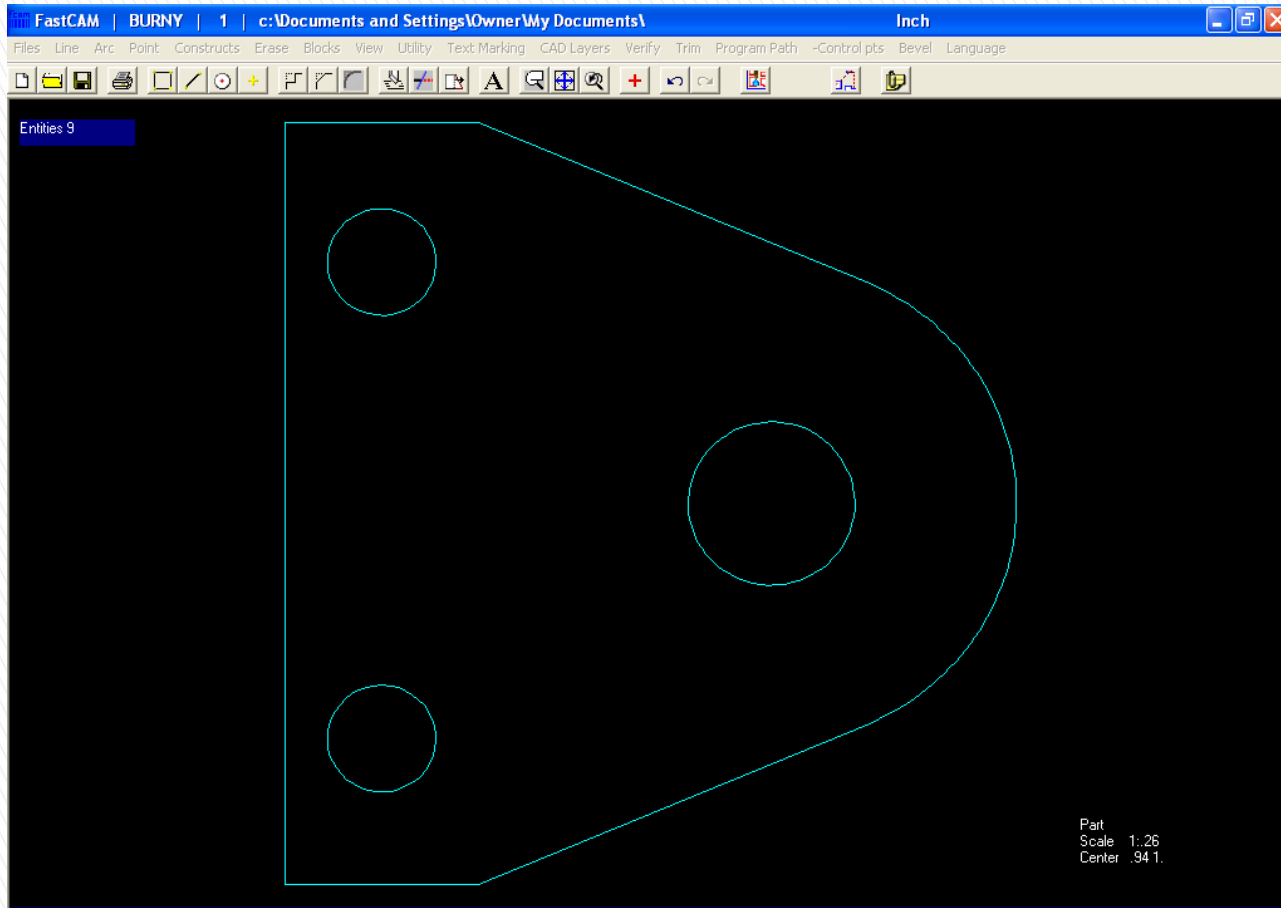
Left click on the line you wish to erase once, and then right click on your drawing to fully erase this entity.

Now we can eliminate the last internal arc that is not part of the final drawing. Go under the trim menu, and select Smart Trim.

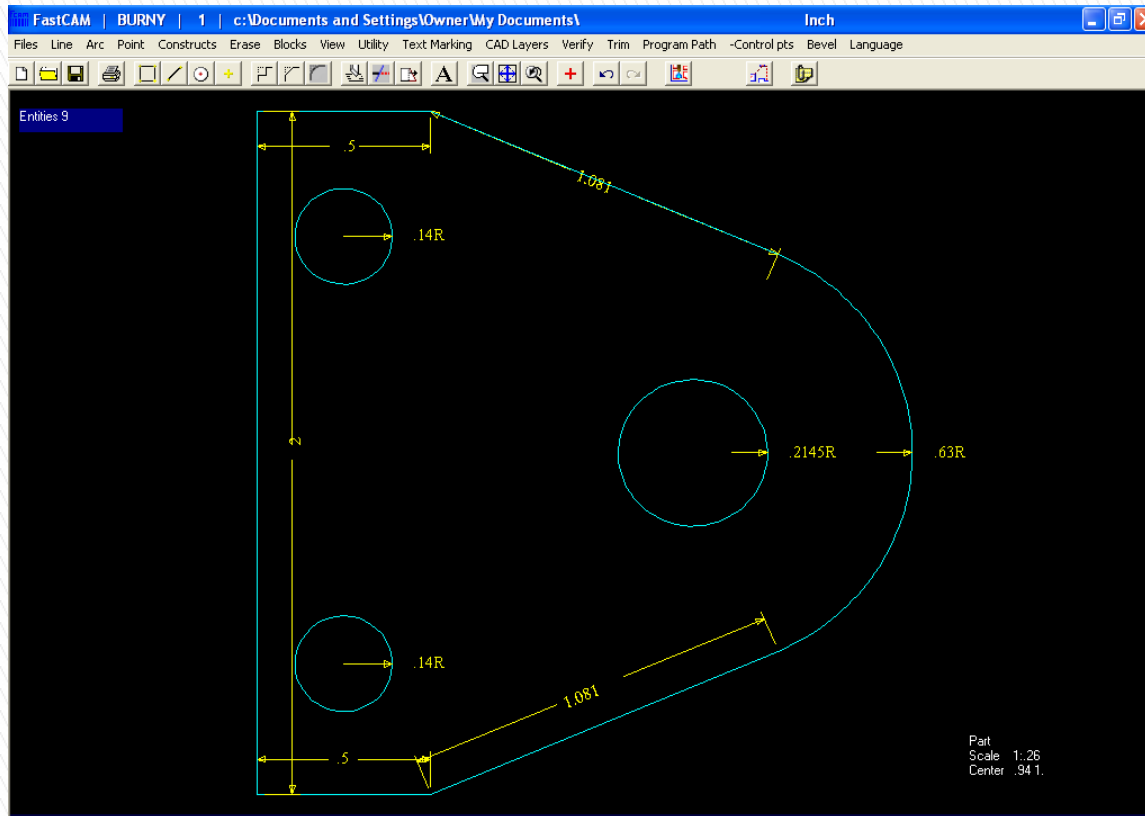


Select the section of the circle that you wish to trim by clicking on the left side of the large circle of the ring that you want to trim. The unwanted arc should be removed.

Now return to the View menu and uncheck the display axes box to remove them from your drawing. You will then see your completed WIDGET drawing.



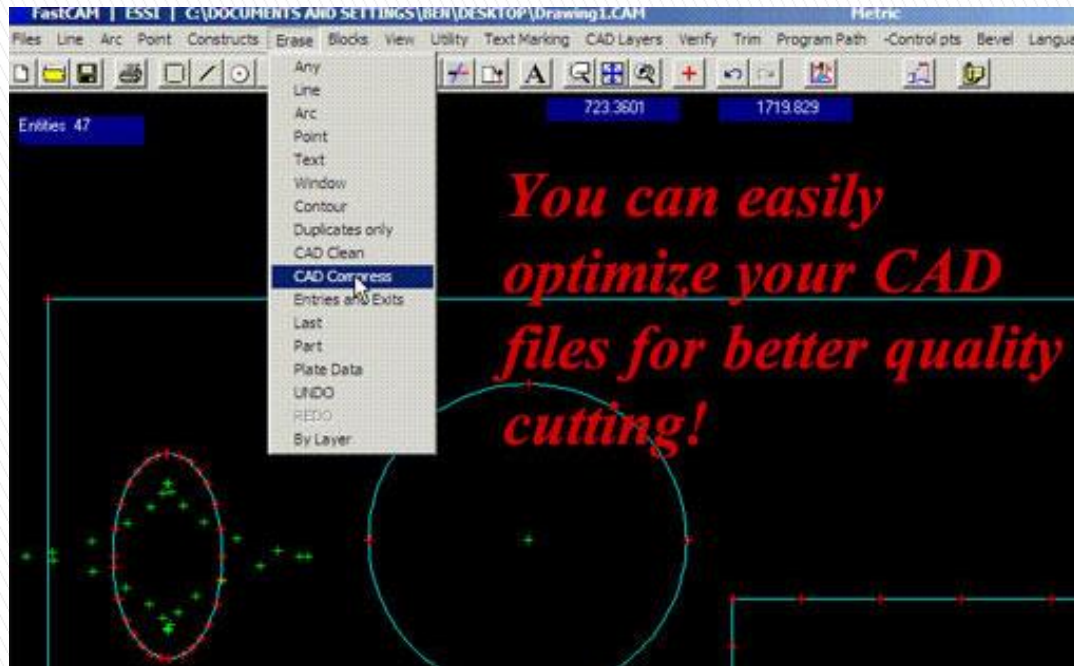
Finally, there are a number of ways to verify the scale of your drawing. The quickest way is to go under the View menu, choose Change display, and check the box next to Auto Dimension. Then click okay to view the dimensions of your part drawing.



We can see our part meets the dimensions of our scaled part drawing. We can be confident that we have created a part with FastCAM® that is identical to our scaled drawing.

# Module 1 Lesson 3

## Working with CAD files (30 minutes)



## 3.1 CAD Optimization

The following CAD features are integrated into the FastCAM Drawing Editor.



Main functions:

- (1) Clean imported CAD files for quality cutting.
- (2) Compress CAD files to improve cutting speed.

Covers: DXF/DWG restore, explode, CAD layers, text marking, dimensions, verify, simple/overcut bridge, fillet corners ... etc.

## **The Importance of Optimization of CAD Clean and Compress**

The easy optimization and compression of CAD data is a valuable feature in FastCAM.

CAD systems are widely used for drawings and DXF/DWG files are often used in NC however they are not always ready for quality cutting. CAD systems often generates redundant points, overlapped lines, small-segment curves and open contours which produces NC output that may cause disordered machine travels, repetitive cutting, nesting problems, programming errors, frequent machine accelerations and decelerations leading to vertical machine jitter, poor cut quality, low productivity and shortened machine life.

FastCAM's CAD clean and CAD compress will fix all this automatically!

Illustrated Cutting Errors:

1. Internal hole double cut.
2. Incorrect position of entry and exit.
3. Part of line cut twice.

Purpose of CAD Clean:

1. Cleaning of all the redundant points to prevent disordered machine travels.
2. Cleaning of the repetitive entities to prevent incorrect entry/exit and double cutting.

**The following examples demonstrate :**

**Clean and Compress CAD files** (Important for all cutting especially LASER)

To optimize the DXF/DWG data, FastCAM has been designed with the ability to automatically clean, compress and verify geometrical entities before they are saved to CAM files.

**Automatic and manual path programming for single part cutting**

NC codes can be generated with both the manual and automatic path setting method for a single part that has to be cut repeatedly.

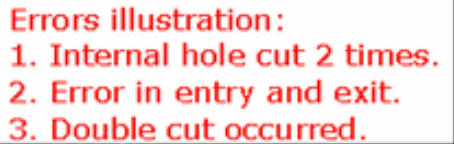
Note: The definition of a Single part can be more than one part contour or a combination of many parts grouped together but they will be processed as a single item.

**Other settings and cutting functions**

'Extract Part', 'Explode Drawing', 'CAD Layers', 'Text Marking', 'Verify Dimensions', 'Simple/Overcut Bridge', 'Fillet', 'Corner to (Fillet, Chamfer)', 'Offset Contour', 'Hole to Point' .... are the common features for CNC LASER cutting.

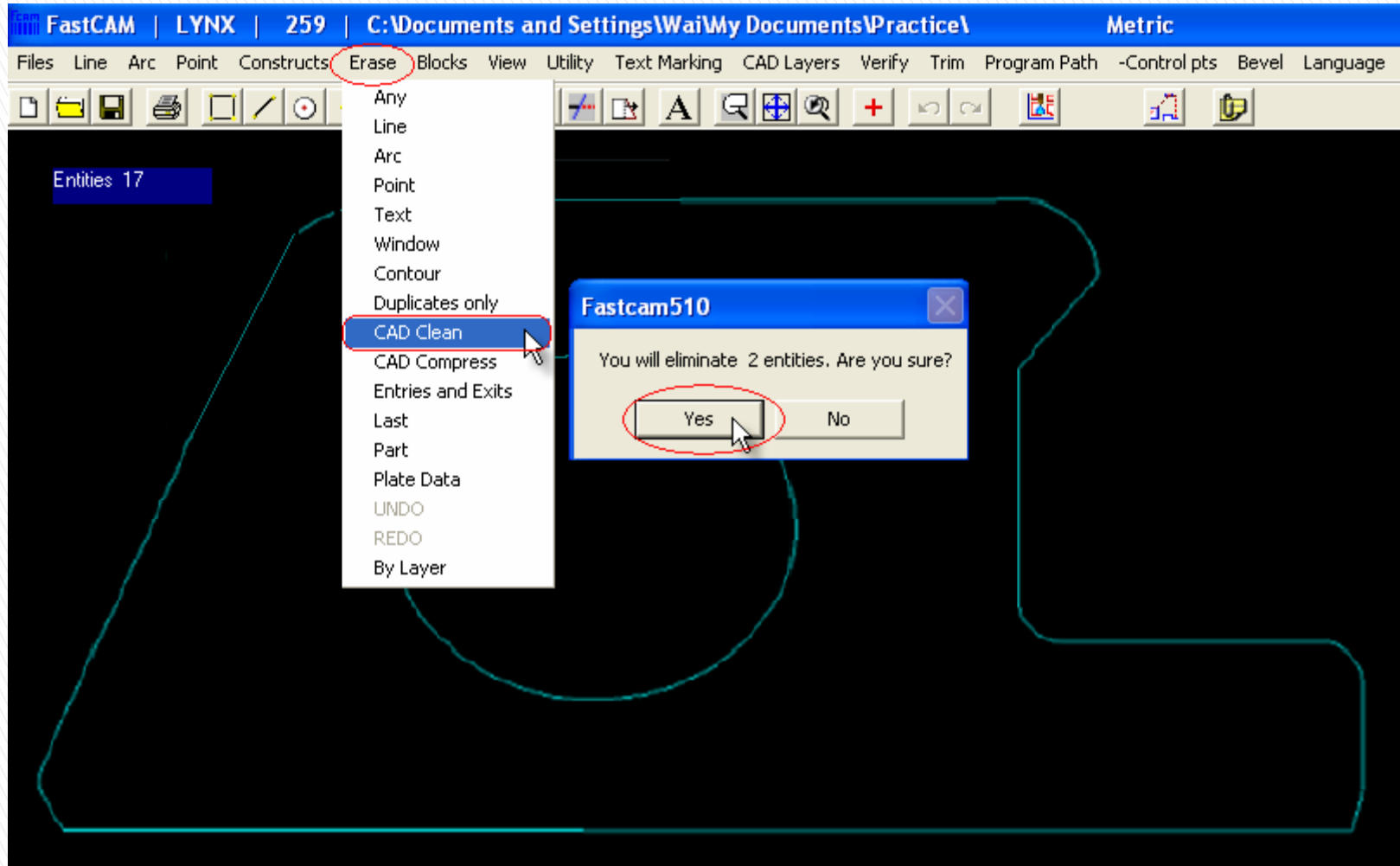


## I. Purpose for CAD Clean



# Lesson 3 CAD Optimization

## II. CAD Clean result



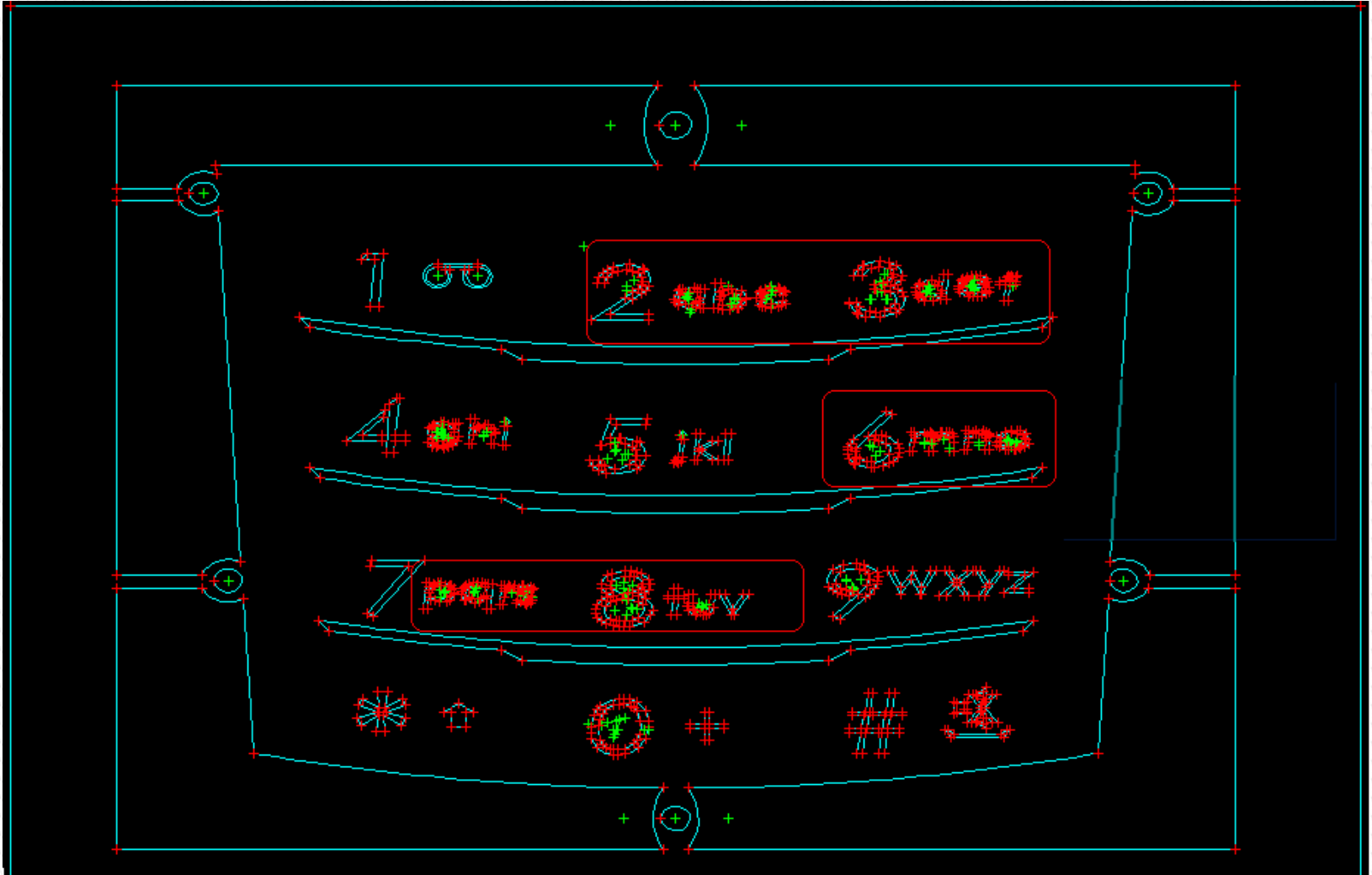
## Brief description:

Total 17 entities. After the procedure of 'Erase' → 'CAD Clean' → Dialog Box (You will eliminate 2 entities. Are you sure?) → (Yes) No, entities reduced to 15 with the case of double cut avoided.

Additionally, closure between curves and contours can be checked with the 'Verify' → 'Contour' function. Any open contour should be closed in the drawing editor, or the speed of nesting in FastNEST will be reduced and may not be able to complete. If the open gap is too small (less than 2mm), FastCAM will automatically take it as closed and the nesting speed will not be affected.

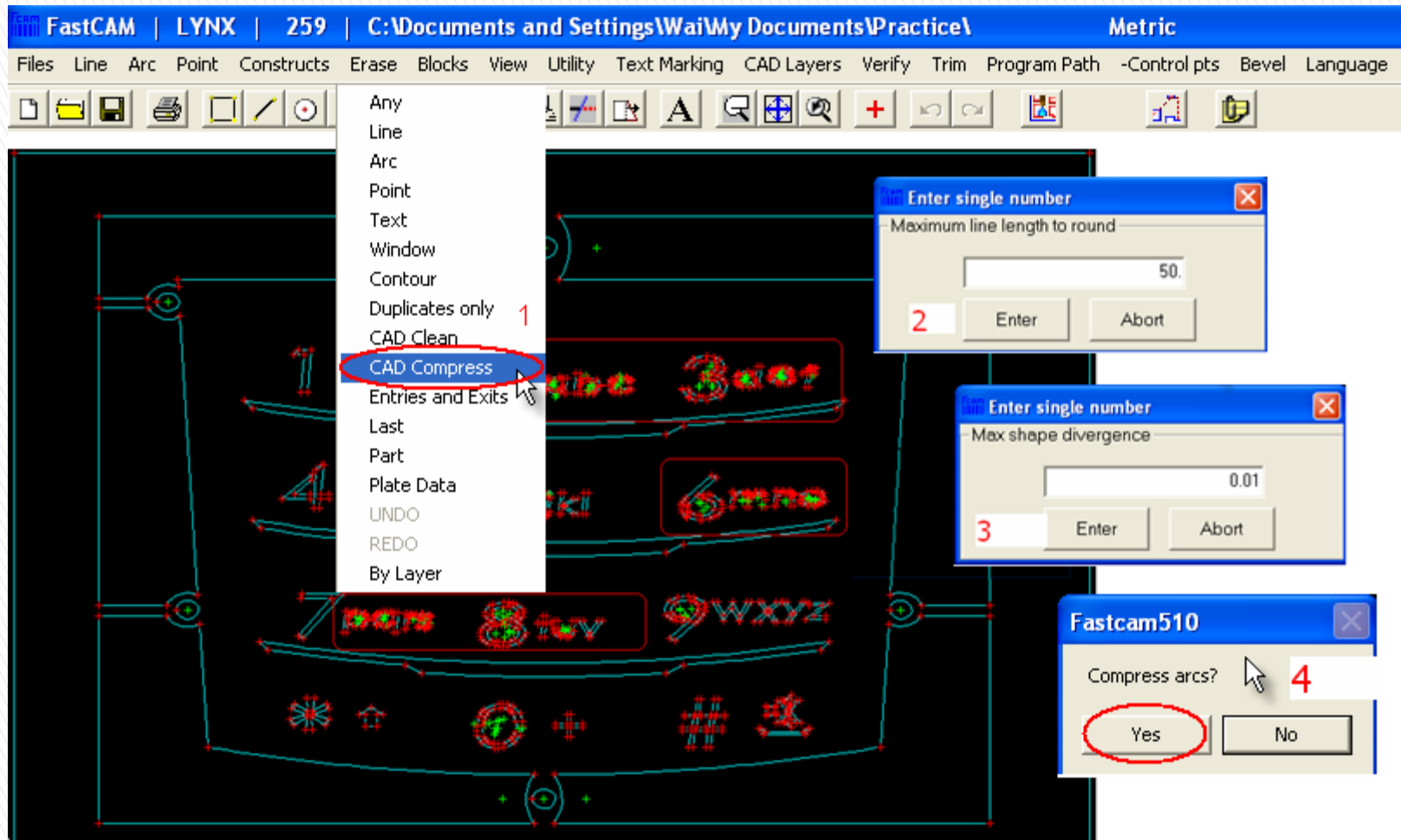
# Lesson 3 CAD Optimization

## III. Purpose for CAD Compress



# Lesson 3 CAD Optimization

## IV. CAD Compress



1270 entities remain after CAD clean but this is not the best number. As observed, there is a lot of red points around the numbers.

#### Description:

As observed, there is a lot of red points around the numbers and many of them are very close to each other. When NC codes are generated under this condition, vibration will occur during cutting which will cause damage to the machine. With CAD Compress, the situation can be improved by reducing the number of entities which is also good for those older controllers with smaller internal memory size.

#### Purpose of CAD Compress:

Compress and fit curves to reduce large amount of short-segment entities, minimize the amount of cutting instructions to avoid machine vibration.

Compress and fit curves to reduce large amount of short-segment entities, avoid open gap between curves and contours, reduce accumulative cutting error for better quality and precision.

### Steps:

ERASE → CAD Compress → CAD Clean? → NO → Max line length to round → ENTER → Max shape divergence (current value 0.01mm) → ENTER (watch for any shape deviation) → Compress arcs? → YES

### Description:

1. Drawing entities reduced from 1270 to 992 after 'ERASE' → 'CAD Compress'. This result can be further improved by applying 'CAD compress' again based on actual requirement.
2. Indicate the max. line length of short segments. to be cleaned/compressed for reduction of program steps and higher cutting efficiency.
3. To avoid or minimize any effect of vibration in LASER cutting, value of 'Max shape divergence' should be 0.01 mm (precision value in file 'SETUP.DAT' needs to be 0.01 mm).

### Useful CAD functions - Extract Part and Explode Drawing

With 'Blocks' → 'Extract Part' and 'Blocks' → 'Explode Drawing' to extract or explode multiple parts from DXF/DWG file for part nesting (typically used for nesting of ship components).

To improve the appearance of sharp corners in LASER cutting, corners can be rounded to fillets (e.g. 0.2 mm) as follows.

'Program Path' → 'Round corners' → 'Corner radius' – 0.2 mm → 'Fillet inside corners?' → 'Yes' → 'Both'

Change to same setting in FastNEST (invalid for NC input file type) –

'Nests' → 'Nest Parameters' → select 'Change corners' → set corner distance, profile type ... and press 'Enter' to confirm.

## **Lesson 3 CAD Optimization**

Exercise - CAD Clean/Compress for DXF/DWG

Exercise 1

- i. Compress number of entities in exercise '1.CAM' from 9871 down to 518 and note there should be no change to its shape.
- ii. Create your company name with a graphics program such as CorelDRAW and save as DXF format. Compress number of entities to minimum without change of shape in FastCAM for normal cutting.

Exercise 2

Check exercise '2.CAM' and find the reasons why this part has more than 18 entities.



## Description:

Learn and master the purpose and method for CAD Clean and Compress by going through 4 exercises.

## Summary of this section:

1. Learn and understand the importance of using CAD Clean and Compress to improve quality, nesting and cutting efficiency.
2. Learn and master the method and skills for CAD compress.
3. Learn and master the method for regular closure and restoration of closure.

## Exercise 1

1. 19 and 0.7 are the 2 parameter values to avoid shape divergence.
2. Correct use of the 2 parameter values for compression.

## Exercise 2

By checking the control points, we can find out what has gone wrong with this parts in showing 33 entities.

5 min. break

# Module 2 NC Tool Path Programming

(45 minutes)

Lesson 3: Manual Path Programming (15 minutes)

Lesson 4: FastPATH Auto Path Programming (30 minutes)

## Tool Pathing

NC path programming has a big impact on cutting speed and quality.

The FastCAM Professional Edition includes the FastPATH module which allows you to automate tool pathing and halve your programming time.

# Lesson 3 Manual Tool Path Programming

The screenshot displays the FastCAM LYNX software interface with the following elements:

- Top Bar:** FastCAM | LYNX | 259 | C:\Documents and Settings\Wai\My Documents\Practice\
- Menu Bar:** Files Line Arc Point Constructs Erase Blocks View Utility Text Marking CAD Layers Verify Trim Program Path -Control pts Bevel Language
- Context Menu (1):** Opened from the 'Program Path' menu, showing options like 'Next Path', 'Delete all paths', 'Delete last path', 'Redefine Path', '\*Output NC code', 'Status', 'Part Description', 'View Path Data', 'View', 'FastPLOT Verify', 'FastPATH', 'Restore Drawing', 'Pathed DXF', 'DELEM DXF', 'Fill Bar', 'Round corners', 'Sharpen corners', and 'Remove Feeds'.
- Entity Label (2):** 'Entities 17' circled in red.
- Marker (3):** 'PLASMA MARKER' label with a cursor.
- Direction Menu (4):** Opened, showing 'Left', 'None', and 'Right' options.
- Inside/Outside Menu (5):** Opened, showing 'Outside' and 'Inside' options.
- Nearest quadrant Menu (6):** Opened, showing 'Nearest quadrant' and 'Indicated point' options.
- Path Type Menu (7):** Opened, showing 'Straight', 'Quarter circle', 'Half circle', 'QC entryexit', and 'HC entryexit' options.
- Radius Dialog (8):** A dialog box titled 'Radius for Entry Exit arcs' with input fields for '10' and '8', and 'Enter' and 'Cancel' buttons.
- Marker (9):** Another 'PLASMA MARKER' label with a cursor.

### Steps:

1. Select 'DXF Restore' from the 'Files' menu and deselect 'Apply Z axis extrusion'. Pick file name from dialog box then select to load the required DXF into FastCAM. Check number of entities from top left of the screen and perform CAD Clean/Compress if it is required.
2. Select 'Program Path' → 'Next Path' → select operation (Cutting/Marker) → select left/right/none for kerf compensation → select contour

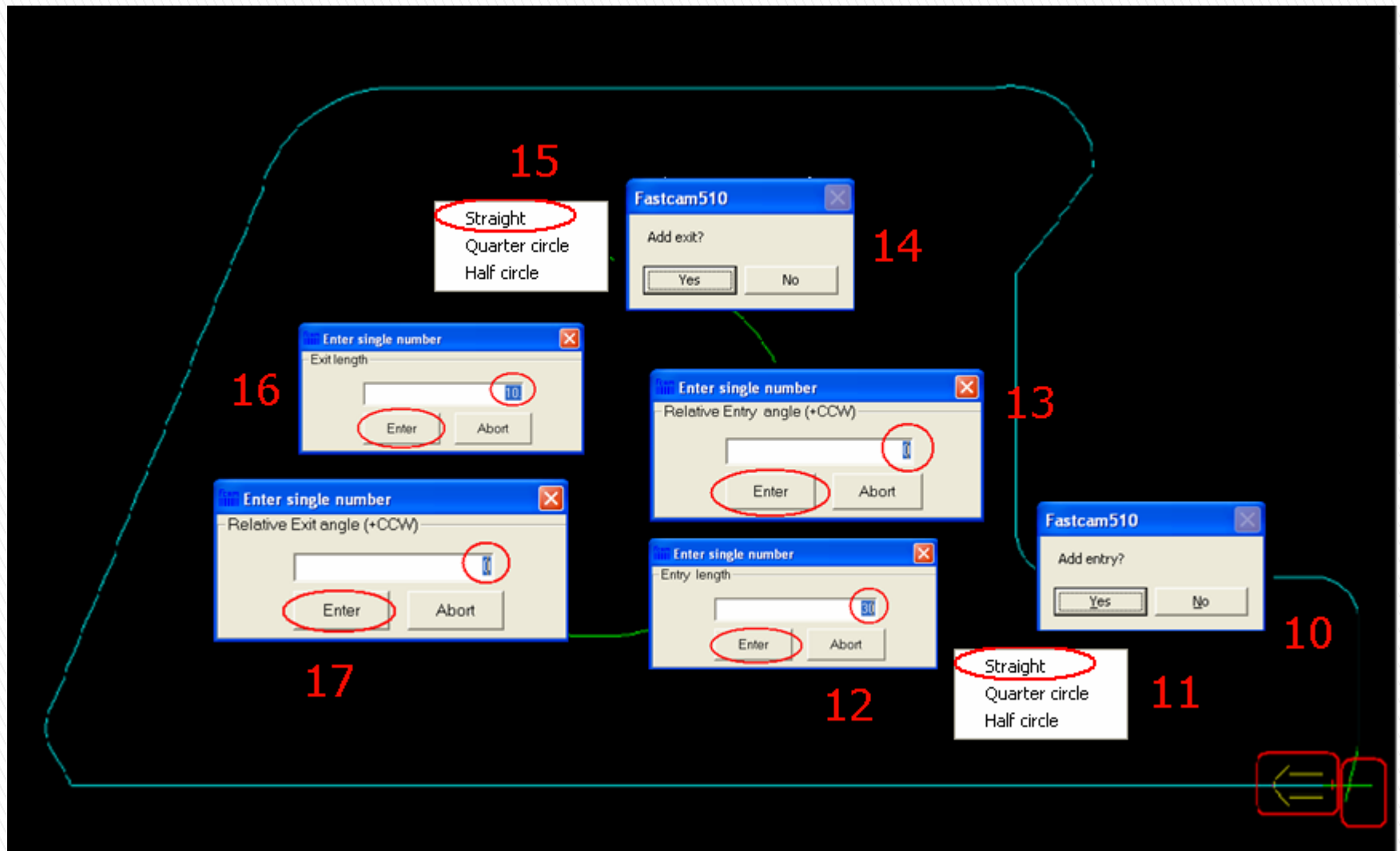
### Must note below principles when doing manual setting:

1. Setting of path parameters for contours starting from internal cuttings to external cuttings.
2. For kerf setting with left compensation, direction of travel must be counter-clockwise for internal contours and clockwise for external contours. For kerf setting with right compensation, direction of travel must be clockwise for internal contours and counter-clockwise for external contours. All contours should be set using the same principle or dimensional error will occur after cutting.

When doing manual setting for cutting paths, it is necessary to pay attention to the cut sequence, direction of travels and angle of entry/exit for both the internal and external contours. This is all about manual setting of the cutting paths and its procedure is actually the same as the automatic option.

To tackle undercut problems occurring on corners during high speed cutting, round corner e.g. 0.2 can be added to paths automatically in FastCAM and FastNEST.

# Lesson 3 Manual Tool Path Programming



Steps: Select 'Program Path' → 'Next Path' → select operation (Cutting/Marker) → select left/right/none for kerf compensation → select contour

These principles apply to any manual setting:

- Setting of path parameters for contours start from internal cuttings to external cuttings.
- For kerf setting with left compensation, direction of travel must be counter-clockwise for internal contours and clockwise for external contours. For kerf setting with right compensation, direction of travel must be clockwise for internal contours and counter-clockwise for external contours. All contours should be set using the same principle or a dimensional error will occur after cutting.



# Lesson 4 Automatic Path Programming

## I. FastPATH

This program is not shown individually on the desktop. You can apply settings in FastCAM or FastNEST.

This program allows you to fully automate the setting of cut paths for optimum operation efficiency with:

- ⑨ Automatic setting of entries and exits.
- ⑨ Set kerf compensation.

You can also set various cutting techniques including common edge, bridging and common cutting.

# Lesson 4 Automatic Path Programming


## 1. Cutting process

**FastPATH Settings** [X]

Help

Process External Entry Internal Entry Cut Sequence Specials Stitch / Kerf Nest Options Bevel Router / Bridges

Selected processes	Kerf	Side	Available processes
PLASMA	Left	Both	
OXY		Both	
MARKER			



**Reset**

**Apply processes**

☒ Across whole nest

☐ Part by part

Select processes in the order required. Change kerf and side by clicking on the item.

**Description** -1

**Accept** **Cancel**

## Selection of machining process

Depending on functions available for your Machine, processes such as CUTTING, MARKER, OXY, PLASMA ... etc can be selected from left of the box by double-click on item or left-click on the left arrow button. If several processes are required (which have not been specially defined), it will be necessary to set individual CAD layer in the FastCAM Editor for each process, e.g. set marking for 1 layer and LASER cut for the other layer. Layers in FastSHAPES will be saved automatically to CAM file but they need to be defined for parts created in FastCAM or other systems. Specific settings can be checked from reference example. Selected processes can be cleared up by left-click on the 'Reset' button. **NB Additional functions such as Plasma , marking etc. must be included in your Control File and supported by your machine. Please contact your supplier or FastCAM Inc. to add these features to the software if required**

## Kerf compensation ( Adjustment for width of cut )

According to operation requirement, single-click to change compensation to left, none or right. Created path will be adjusted according to this kerf compensation setting and the default compensation is **kerf left**.

## Side ( especially important when processing previous nests with a plate boundary or parts within parts)

Double-click here will select which side(s) of contours has to be processed - inside, outside or both in addition to the kerf compensation. With this function, it is possible to define different processes for internal and external contours, e.g. to set paths of internal contours (inside) first for Oxy cutting and external contours (outside) for plasma cutting. For general applications, side has to be set to 'Both' with the same process for both internal and external contours.

## Available processes

Here is a total list of the FastCAM available processes which have not been set to FastPATH application. They can be selected for FastPATH settings by double-clicking on the process or left-clicking the process and then the left arrow. Selections can be cleared by left-click on the 'Reset' button.

## Apply processes

Across whole nest – this option allows the processing of all nested parts according to the defined sequence, e.g. marking for all parts followed by cutting typically. With this operation model, distance of rapid travel can be shortened and time can be saved from frequent changing of processes.

Part by part – this option allows the operation of selected processes one by one for a single part such as marking followed by cutting for one part until all parts are processed which is commonly used in LASER cutting.

## Meaning for cutting

Oxy cutting is a process of preheating steel to its combustion temperature, then burning it rapidly by means of a regulated jet of oxygen with a gap (known as kerf) left behind. Plasma cutting uses the same principle but thermal energy for heating of steels to combustion temperature is converted from electrical energy.

Compensation of kerf is necessary for parts to be with correct dimensions and is defined to be left or right about the line of contour as centre.

For kerf setting with left compensation, direction of travel must be counter-clockwise for internal contours and clockwise for external contours. For kerf setting with right compensation, direction of travel must be clockwise for internal contours and counter-clockwise for external contours. All contours should be set using the same principle or dimensional error will occur after cutting.

# Lesson 4 Automatic Path Programming

## 2. External entry

**FastPATH Settings**

Help

Process External Entry Internal Entry Cut Sequence Specials Stitch / Kerf Nest Options Bevel Router / Bridges

Position: Top right

**Entry**

☒ On/Off      Type: Straight

☐ Corner pierce      Length: 10 mm

☐ Extension      Angle: 0 deg

☐ Length: 0

**Exit**

☐ On/Off      Type: None

Length: 0 mm

Angle: 0 deg

Set your Lead-in and Lead-out preferences.

**FastPART**

☐ On/Off      Height: 6 mm

Width: 2 mm

Max Length: 0 mm

Plate starts: None

Opportunity: Overcut

**KERF**

☐ Single (plasma)      ☒ Both (Oxy)

**Movement minimization**

☐ Columns only      ☐ Dual kerf cut

☐ Rows only

**Extras**

☐ Avoid protected layers

☐ Plate edge breakup

Description: -1

Accept Cancel

**Entry**

Piercing is normally required to start cutting and the size of the pierced hole is always bigger than the kerf. That means the part will be damaged if piercing is started right on the contour which is obviously not desired. For this reason, piercing has to be started away from contours with a line known as an entry.

**Exit**

To ensure completeness of contours and separation of parts after cutting, an extended line known as the exit is normally cut at end of contours.

**Position**

This drop down is for selecting the position of entry and exit on parts. Available selections are Top left, Bottom left, Bottom right, Top right, Top, Right, Bottom and Left. If an obvious corner cannot be identified, the most appropriate position will be selected by the program which may be different from what you want. To prevent its happening, a point has to be created as the entry on contour. Entrance point will be added at mid point of the first entity if no corner has been selected.

Suggestion: To reduce thermal distortion, position of entry at Bottom right is suggested if the zero point of your machine is top left.

**Corner pierce**

Entry will start at corner of contours if this is selected. Otherwise, entry will start from mid point of the closest entity.

**Type**

Type of entry can be straight line, quarter circle, half circle or none. You can also manually select these within FastCAM.

**Extension**

This option will be automatically activated if entry with quarter circle is selected. This is a new function developed to solve the undercut problem in LASER cutting due to fast cutting speed. It can be useful to PLASMA cutting also.

**Length**

To specify length or radius of the entry here. Unit of input will depend on the FastCAM setting with either a fraction or decimal value.

**Angle**

Input here for angle of the entry at 0, 45 or 90 degree. Straight line entry on arc contour will be appropriately created if selected angle is not practical.

**On/Off**

For selection of entry and exit.

# Lesson 4 Automatic Path Programming

## 3. Internal entry

**FastPATH Settings**

Help

Process External Entry **Internal Entry** Cut Sequence Specials Stitch / Kerf Nest Options Bevel Router / Bridges

☐ Internal same as External

**Position**

- ☐ Pierce in Corner
- ☐ Break
- ☒ Break longest entity

**Entry**

☒ On/Off Type: Quarter Circle  
Radius: 3 mm  
Angle: 0 deg  
☐ Extension Length: 0

**Exit**

☒ On/Off Type: Quarter Circle  
Radius: 2 mm  
Angle: 0 deg

**Center Pierce**

Center pierce if radius < 3 mm

Description: -1

Accept Cancel

**Internal same as External**

Entry and Exit of internal contours will be set to same as those for external contours automatically once this box is selected.

**Pierce in Corner**

All entries will start from corners closest to the external entries once selected. Closest point will be used if the selected corner does not exist.

**Break longest entity**

The longest entity of internal contours will be selected and broken at the mid point where entry and exit will be added. This function is suitable for complex rectangles or similar parts.

**Center Pierce**

This option enables small holes to be pierced at the hole centre if the hole radius diameter is under a certain value. This allows correct piercing of small holes without changing the default value for length of entry.

For example, the smallest hole that can be pierced correctly will be  $\frac{1}{2}$ " (12.7mm) in diameter if length of entry is set to be  $\frac{1}{4}$ " (6.35mm) – equal to radius of the hole if entry starts from centre.

By specifying a value here, any hole with its radius less than the value will be centre pierced with the entry ignored. However, there will be no effect to the current FastPATH setting or piercing for small holes if this radius is set to zero.

**Type**

Type of entry can be straight line, quarter circle, half circle or none. All of these can also be selected for manual setting in FastCAM Editor.

**Length**

To specify length or radius of the entry here. Unit of input will depend on the FastCAM setting with either fraction or decimal values.

**Angle**

Enter the angle of the entry at 0, 45 or 90 degree. Straight line entry on arc contour will be appropriately created if selected angle is not practical.

**On/Off**

Turns on or off selection of entry and exits.

**Extension**

This option will be automatically activated if entry with quarter circle is selected. This is a new function developed to solve the undercut problem in LASER cutting due to fast cutting speed. It can be useful to PLASMA cutting also. Extension adds a small straight line to arc entries.

# Lesson 4 Automatic Path Programming


## 4. Cut Sequence

**FastPATH Settings** [X]


Help

Process External Entry Internal Entry **Cut Sequence** Specials Stitch / Kerf Nest Options Bevel Router / Bridges

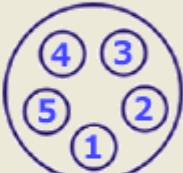
Internal Sweep Columns [v] Choose cutting sequence for Internal penetrations.



Rows



Columns



Next Closest

☐ Preserve original sequence ☐ Aligned with next pierce

Nest Start Bottom left [v] Choose the Start Position for the Nest, along with the nest's cutting sequence.

Nest Sweep Columns [v]

Description -1 [ ]

Accept Cancel



## **Cut Sequence**

To specify the sequence for processing parts in a nest. It can be in the order of Columns, Rows or Next closest. NB The direction of the sequence depends on the Origin point of the nest (Bottom Left , Top Right etc.)

## **Nest Start**

To define location of the first reference point of the nest. This is normally set to bottom left but can be top left, bottom right or top right and is applicable to single or multi-plate nesting.

## **Nest Sweep**

Sets the priority for the cutting sequence in nesting when you set the sequence as Next Closest or Minimum Rapid This can be from left to right or bottom to top and return following the specified direction.

# Lesson 4 Automatic Path Programming

## 5. Special Setting


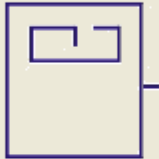
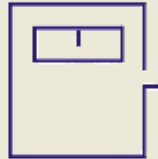
**FastPATH Settings**

Help

Process External Entry Internal Entry Cut Sequence **Specials** Stitch / Kerf Nest Options Bevel Router / Bridges

**Tag**

☐ On Length  mm Tag Contours

All Contours Inside Only Outside Only

**Small Hole**

Maximum Size  mm

Feedrate  %

**Entry**

Feedrate  %

**Pre-Pierce**

☐ On/Off Radius  mm

Cuts hole of specified Radius, at all profile entry positions; prior to commencing the part cutting. All Pre-Pierces will be completed before any profile is cut.

Description

Accept Cancel

### **Tag (final uncut part of any contour)**

Tag can be used to prevent parts dropping, curling up of parts and overburn of internal holes during cutting. It also can be used to eliminate or minimize the occurrence of part contours (particularly holes) being damaged from notches created by entry/exit of paths.

By turning on this option, type of contour to be tagged and its length can be entered.

### **Pre-Pierce ( pre burning of pierce points across a nest, normally heavy plate cutting with Oxy )**

For initial processing of parts across the whole plate. Pre-piercing of holes is often done with an oversize or worn out nozzle before part cutting. For Laser and Plasma cutting, piercing can also be done at much higher values than for cutting. Once piercing is complete the machine is set to the correct values and the precut pierces used to complete the nest at optimal speed and quality. Pierce times should be reset to allow "edge starting" on the pre cut holes.

When using this feature, it is necessary to include appropriate codes in the FastCAM control file (e.g.

EDGE.CON ... \*.CON)

/PREPIERCE/ //This is the command for pre-piercing.

M07 //Cutting start

G01Y5.0 //Cut 5mm above the pierce hole

G03J-5. //Circular interpolation

M08 //Cutting end

G00Y-5. //Return to original pierce location.

Note: Remarks before // are not required, delete these codes if this function is no longer required.

### **Small Holes (Automatic slowdown of cutting speed when processing small holes)**

Drag in the cutting process creates a bevel effect when cutting curves or holes this is especially noticeable when cutting small holes I.E. the smaller the hole the greater the effect. Automatic reduction of the machine feed rate to a certain percentage, e.g.  $2000 \times 70\% = 1400$  for small holes markedly improves cut quality  
NB Especially useful on heavy plasma cutting.

**NB Test cutting will determine ideal settings for small holes and entries.**

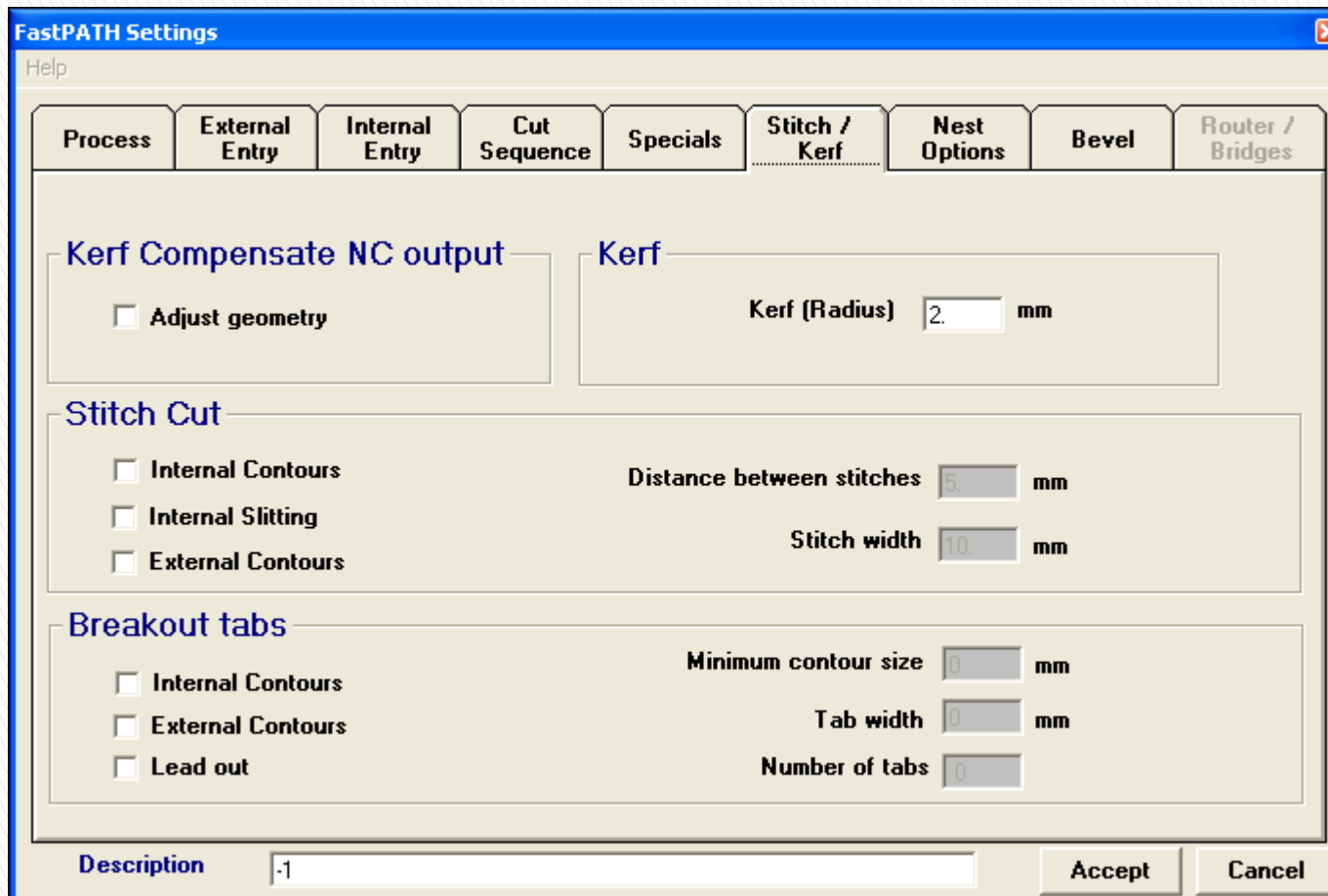
### **Entry**

Adjustment of feed rate for entry to start cutting.

**The function of feed rate reduction for small hole and entry is an important processing technique to achieve good quality of small hole, arc and thick contours in HD Plasma and LASER cutting.**

# Lesson 4 Automatic Path Programming

## 6. Stitch / Kerf Compensation



The image shows the 'FastPATH Settings' dialog box with the 'Stitch / Kerf' tab selected. The dialog has a blue title bar and a 'Help' button. The tabs are: Process, External Entry, Internal Entry, Cut Sequence, Specials, **Stitch / Kerf**, Nest Options, Bevel, and Router / Bridges. The 'Stitch / Kerf' tab contains three sections: 'Kerf Compensate NC output' with an unchecked 'Adjust geometry' checkbox; 'Kerf' with a 'Kerf (Radius)' input field set to 2 mm; 'Stitch Cut' with three unchecked checkboxes ('Internal Contours', 'Internal Slitting', 'External Contours'), a 'Distance between stitches' input field set to 5 mm, and a 'Stitch width' input field set to 10 mm; and 'Breakout tabs' with three unchecked checkboxes ('Internal Contours', 'External Contours', 'Lead out'), a 'Minimum contour size' input field set to 0 mm, a 'Tab width' input field set to 0 mm, and a 'Number of tabs' input field set to 0. At the bottom, there is a 'Description' field with '-1', and 'Accept' and 'Cancel' buttons.

**FastPATH Settings**

Help

Process External Entry Internal Entry Cut Sequence Specials **Stitch / Kerf** Nest Options Bevel Router / Bridges

**Kerf Compensate NC output**

☐ Adjust geometry

**Kerf**

Kerf (Radius) 2 mm

**Stitch Cut**

☐ Internal Contours ☐ Internal Slitting ☐ External Contours

Distance between stitches 5 mm

Stitch width 10 mm

**Breakout tabs**

☐ Internal Contours ☐ External Contours ☐ Lead out

Minimum contour size 0 mm

Tab width 0 mm

Number of tabs 0

Description -1

Accept Cancel

## **Kerf compensate NC output - Adjust part geometry (this box must be ticked to activate kerf compensation)**

Kerf compensation is necessary in producing correctly cut dimensions. This function is used where the programmer wishes the compensation to be done within FastNEST by adjusting part size, not at the machine. This can significantly speed up program time on many older controllers (EG Burny)

**NB Users must ensure accurate Kerf settings are used here, test cutting is advised if unsure.**

### **Kerf**

Enter value greater than 0 here if kerf compensated NC output is required. When this setting is in effect, G41 or G42 will not appear in the NC output.

### **Stitch Cut**

In situations such as the cutting of a long, thin rectangular piece, large distortion can be result from a single complete cut of the contour. To minimize this problem, single or multiple sections of the path will be left uncut, creating the link(s) to hold part in position whencutting.

### **Breakout tabs.**

This feature leaves small breaks in the cut boundary to retain the part in the plate. This is very useful recovering small parts on completion of the nest.

Note: **Above functions can also be applied to LASER cutting.**

# Lesson 4 Automatic Path Programming

## 7. Nest Options

**FastPATH Settings**

Help




Process External Entry Internal Entry Cut Sequence Specials Stitch / Kerf **Nest Options** Bevel Router / Bridges

☐ **Hole Avoidance** Use to avoid traversing over previously cut-parts.  
☒ Rectangular rapids ☐ Straight rapids

☐ **Sort Marking** When marking is defined it may not be optimized. This option finds marking paths and minimizes travel.

Contour Gap  mm Maximum gap allowable in parts before re-piercing.

Outline shown  Alters the cut parameter of the outer and inner contours, as shown below.

None Nest Plate Silhouette

☒ **Cut Short Rapids**  
Distance of Cut short

Description

## **Hole Avoidance**

This function is mostly used in Plasma and LASER cutting because cut parts can move or curl up and obstruct the torch. Since the cut torch will automatically lower during its movement, its collision with the cut parts can occur during travel over the cut areas. By turning on this function, rapid transverse of the cut torch over cut items can be avoided.

## **Sort Marking**

Re-arrange the sequence of markings to improve productivity by reducing the time taken to switch between processes.

All marking is moved to the start of the program followed by the cutting process improving productivity.

## **Contour Gap**

Special fault tolerance can be accepted by FastCAM which is for non-closed contours restored from DXF/DWG/CAM files that cannot be identified with the naked eye. By specifying the maximum allowable gap, the software can automatically process the gap as a closed contour.

## **Outline shown**

Select different cutting methods for different applications.

None: default value to cut both the internal and external contours.

Nest Plate: no cutting for edges of plate, cut for external contours of internal parts and keep them.

Silhouette: cut for internal contours of internal parts and keep external contour of the plate.

## **Cut Short Rapids**

Use in the cutting of thick plates to reduce hole piercing. Where the end of one part is close to the beginning of the next the process continues cutting. Set the value to a suitable distance based on the nest and material.

# Lesson 4 Automatic Path Programming

## 8. Router / Bridges

**FastPATH Settings**

Help

Process	External Entry	Internal Entry	Cut Sequence	Specials	Stitch / Kerf	Nest Options	Bevel	Router / Bridges
---------	----------------	----------------	--------------	----------	---------------	--------------	-------	------------------

**Router**


Depth on each pass  mm


**Bridges**


Bridge Type


Bridge Width  mm

Entry Length  mm

**Bridge** 

**Negative Bridge** 

**Skip** 

**Skip + Entry** 

**Description**

**Accept** **Cancel**



**Router** (this function is under development)

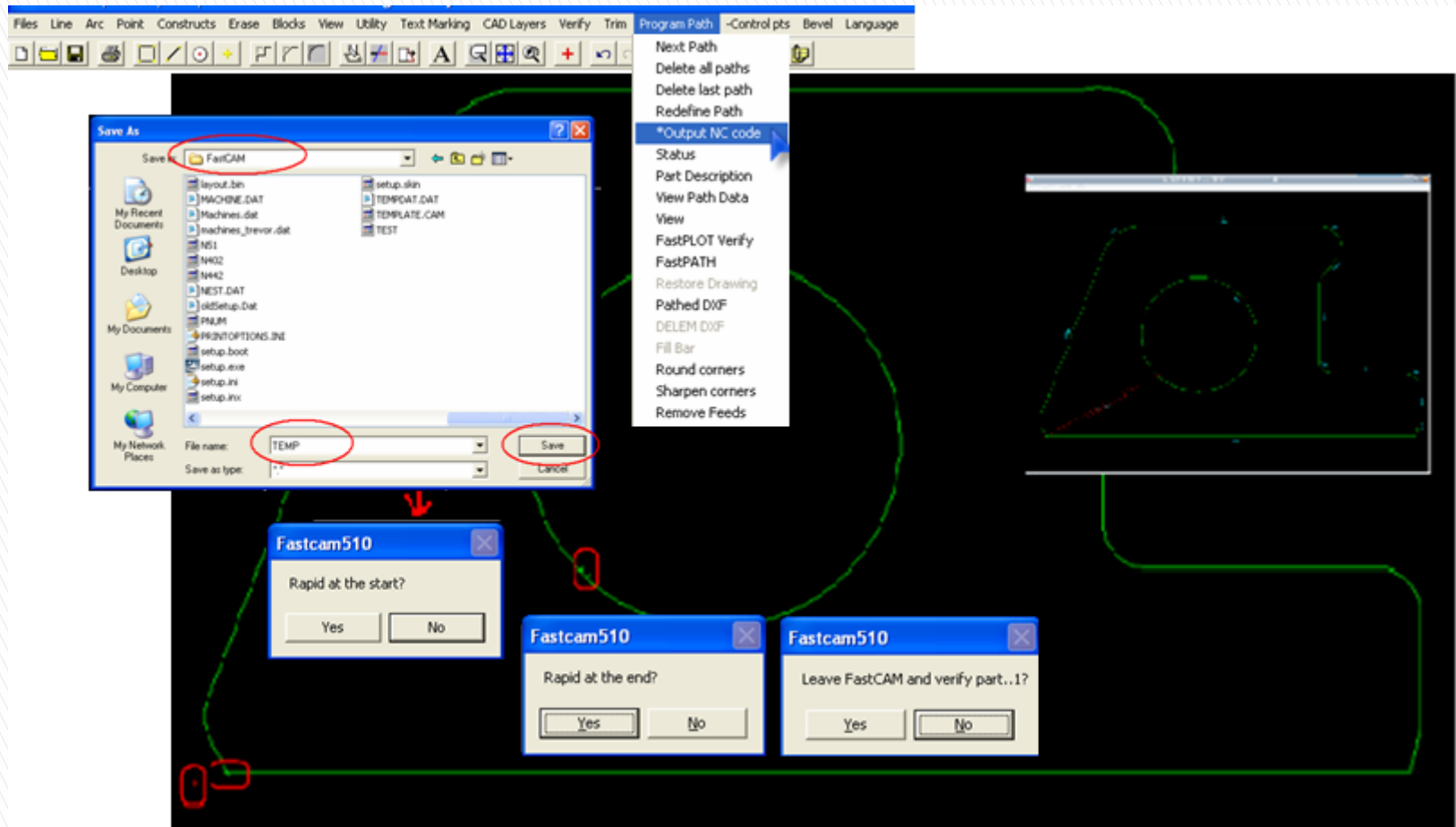
## **Bridges**

To minimize the wear and tear on consumable items in Plasma or other cutting processes, the number of pierce holes can be reduced by using bridge or negative bridge to effect continuous cutting of individual parts as an entire item. Under normal conditions, the minimum bridge width should be larger than the kerf value.

# Lesson 4 Automatic Path Programming

## II. FastCAM automatic path programming

### 1. FastCAM manual output of NC codes (single part)



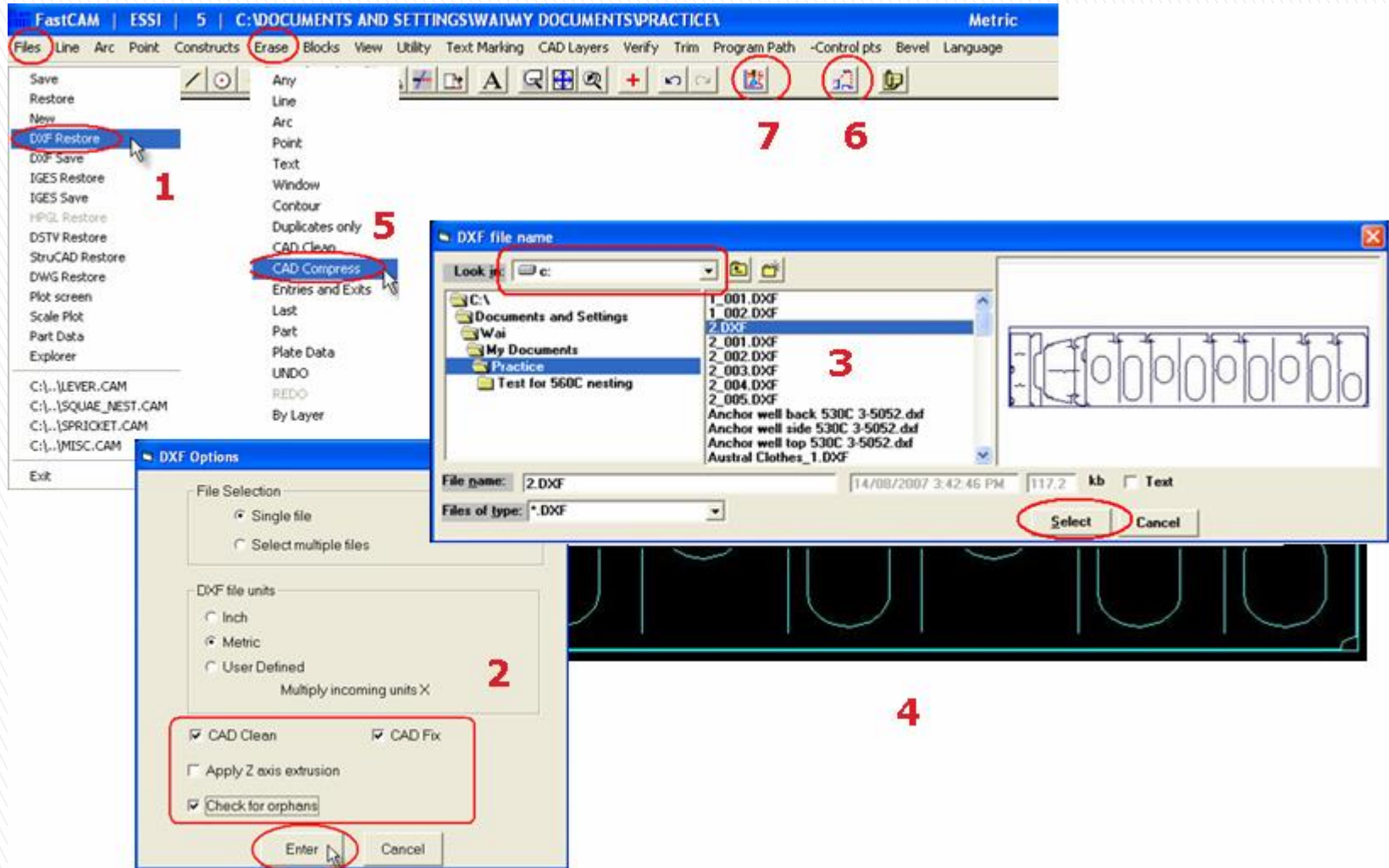
Using the FastCAM drawing editor, DXF/DWG files can be cleaned and compressed for CAD optimization or new drawings can be created from scratch. Manual programming can be managed directly through the menu of Program Path, followed by Next Path. Or you can use the FastPATH module to manage and generate NC output automatically based on the settings selected. Note: FastPATH Settings can be saved for all sorts of materials and processes and restored when required.

Automatic output method:

After path setting through the manual or automatic method, left-click on the 'Output NC code' button. A dialog box will pop up with 'Save as' file name. Then there will be the option to select Rapid at the start? and Rapid at the end?

# Lesson 4 Automatic Path Programming

## 2. FastCAM manual output of NC codes (nested parts)



Automatic path setting and NC output for nested DXF/DWG files are very practical functions available in FastCAM, particularly applicable to the ship building industry. With nested DXF/DWG files prepared by ship design departments, NC codes for different machines can be generated directly in FastCAM for cutting. Nested DXF/DWG files can also be converted into part drawings for filing together with the processing documents.

Steps:

Click on FILES

→ DXF RESTORE or DWG RESTORE

→ un-tick box for 'Apply Z axis extrusion' because general cutting machines do not have the 3rd axis

→ must select box for CAD CLEAN, CAD FIX and CHECK for ORPHANS

Notes:

1. DWG restore is not supported for AutoCAD files prepared with version 2006 or above, please use 'save as DXF' in AutoCAD to prevent the possible error.
2. Select location of file, file type and file name from the dialog box for loading part into FastCAM.
3. For creation of cutting paths, all process parameters have to be set in FastPATH first and then click on the Output NC Code button. NB Save settings for future use
4. Type in file name and this will be the file sent to machine for cutting.
5. If required select yes to 'Rapid at the start?' and pick a point on screen which is normally the intersecting point of the line extensions for length and width of the part – equivalent to a corner of the plate for easy point definition and correct cutting of parts.
6. Select yes to 'Rapid at the end' for return of torch back to start point after cutting.
7. By answering yes to 'Leave FastCAM and verify part ... ?', FastPLOT will be called for NC codes generation and verification with their cutting and travel paths displayed on screen.
8. Single click on screen will enter the FastPLOT NC Editor. By selecting VIEW, PLOT UP TO and trace ON, the execution of NC commands and a step-by-step illustration for cutting can be seen by keep clicking left button of the mouse or space bar of the keyboard.

# **Lesson 4    Automatic Path Programming**

## **3. FastPLOT Verification**

Main functions:

### **1) Cutting path verification**

Simulate the NC file, step-by-step trace on and cut dimensions.

### **2) Cost audits**

Audit cost of material and cutting according to the programmed result.

### **3) NC file and codes interchange**

Convert NC codes back to part drawing or change between NC codes.

## The Value of NC simulation:

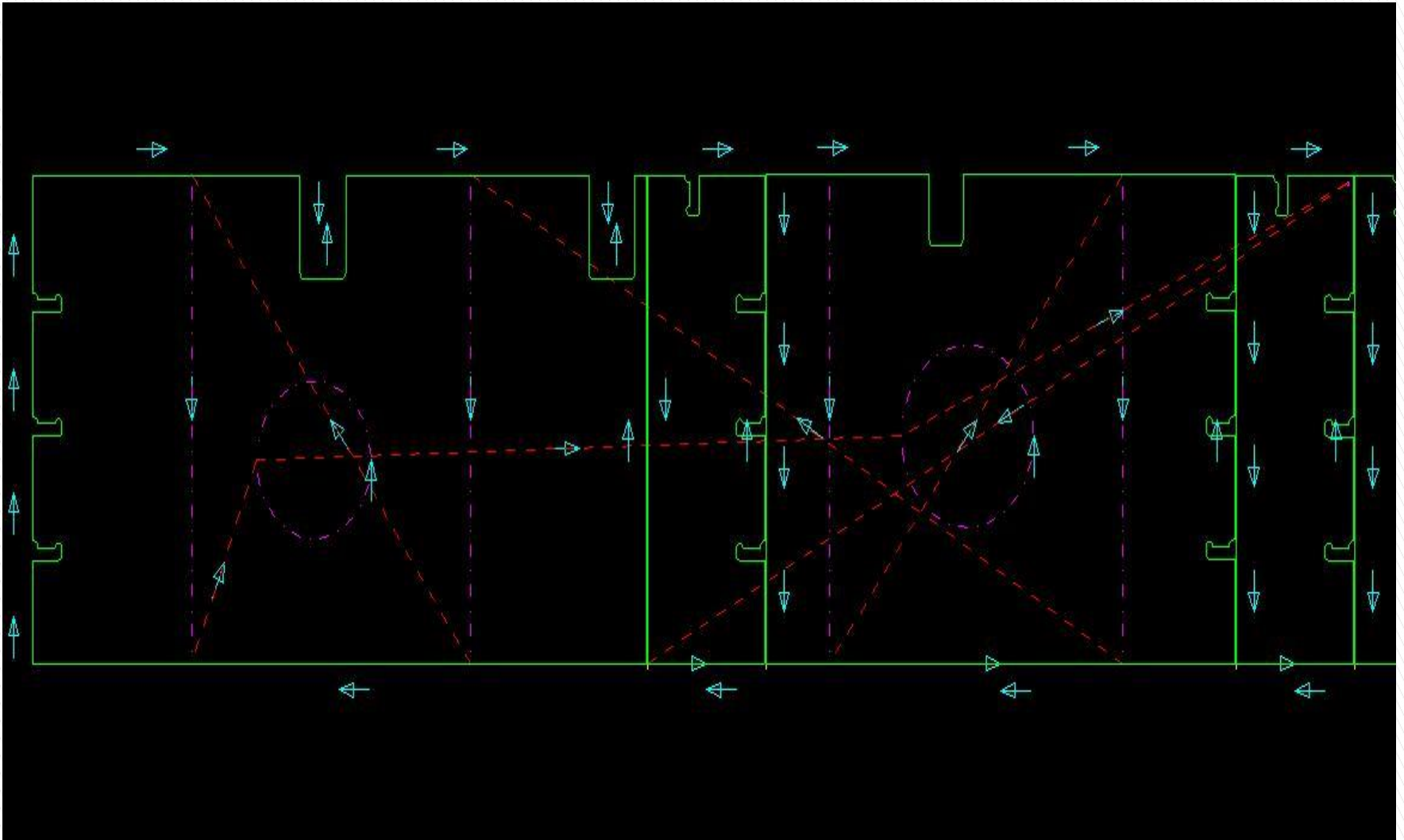
To avoid costly reworks, verification and simulation is an important function.

Problems in cutting and related techniques can be solved quickly through the verification of cutting paths. The added benefit is that you can obtain a simple cost estimate.



# Lesson 4 Automatic Path Programming

## 1) Cutting paths verification





In FastPLOT NC editor, select VIEW → PLOT UP TO → trace ON

Execution of NC commands and a step-by-step illustration for cutting can be seen by keep clicking left button of the mouse or space bar of the keyboard.

Key 'ESC' can be used to abort plot and screen will return to the NC editor after execution.

By selecting Plot Menu from View, diagram can be zoom in, zoom out, show distance ... etc.

# Lesson 4 Automatic Path Programming

## 2) Cost audits

FastPLOT | ESSI | 5 | C:\Documents and Settings\WaiWai Documents\Practice\NC463 Metric

NC File NC Utilities NC Profile Language

### NC Profile Costing

Print Exit

Filename NC463 Date 11-09-2007 Time 11:33:52

Blocks	0	Part Thickness	5 mm	Rect. Width	5946.2 mm
Characters	4402	Material	MILD-STEEL-250 GR250	Rect. Height	1169.8 mm
Machine	ESSI	Density	7850 kg/m3	Rect. Weight	273.018 kg
Control	LYNXESSI.C	Cut Area	5.39 m2	Utilization	77.48 %
Torches	1	Cut Weight	211.532 kg	Scrap Weight	61.486 kg

Process	Distance	Feedrate	Pierces / Starts	\$/Pierce	Secs/Pierce	Time (Minutes)
RAPID	48.17		1.			0
PLASMA	40.76	3000	23.		20	13.97
MARKER	0		1.			0

+ Handling	30 Minutes	Pierce Costs \$	460.
Total Time	43.97 Minutes @	120 \$/Hour = Machine Time \$	87.94
Weight	.273 Tonnes @	900 \$/Tonne = Material Cost \$	245.72
Total \$			793.66

FastPLOT includes a complete cost calculation for part processing.

Method:

Select NC File → Cost → Select NC file for calculation from dialog box → type of material and enter the necessary data for FastPLOT to calculate the machine time, material and total cost.

For example, items red circled in table have to be filled in for calculation to complete.

# Lesson 4 Automatic Path Programming

## ▶ Manual Path Programming Exercises

1. Use the manual path method, generate NC outputs for exercise "1.CAM" after the CAD optimization and study the result.
2. Use the manual path method, generate NC outputs for exercise "2.DXF" after the CAD optimization and study the result.

Requires: set process for cutting, entry with straight line at 8 mm length, angle equals zero and no exit. Also try the half/quarter circle and 90 degree angle for entry to observe the difference in results.

## ▶ Automatic Path Programming Exercises

1. Use the automatic path method, generate NC output for part "1.DXF" after the CAD optimization and study the result.

Set: Straight line for external entry and exit with length of 8 and 6 mm, quarter circle for internal entry and exit with 6 mm radius, 10.1 mm for center pierce hole, no overcut, no compensation and follow the default values for the rest.

2. Use the automatic path method, generate NC output for part "2.DXF" after the CAD optimization. Verify NC outputs in FastPLOT, check any change in dimensions and explain any difference.

Set: Straight line for external entry and exit with length of 10 and 6 mm, quarter circle for internal entry and exit with 8 mm radius, no overcut, compensation at 1.8 mm, nest plate for nest options and follow the default values for the rest.

# Benefits of Automatic Pathing in Review:

FastPATH saves programming time by automatically determining the best cutting paths based on the settings that you apply. FastPATH has artificial intelligence which means it automatically determines the inner-most contours and sets them as the first cutting paths. It also has the ability to determine which holes to cut first to minimize torch movement. But most importantly, it can determine which type of lead-ins and where to place them on the profile of your parts. Blended entries and exits are added automatically.

Complicated parts that used to take upwards of 30 minutes to path out manually only take seconds when using FastPATH. Kerf, cut direction, internals and externals are determined automatically. Path complex parts or an entire CAD nest with a single click of your mouse!

For multi-process systems, you can even select the order of processing. FastPATH will save your settings for different plate styles. Obviously your lead-ins for 4 inch thick plate are going to be different than 1/4 inch plate. FastPATH will allow you to save and restore settings for various styles of plate with just a few mouse clicks.

5 min. break

## Module 3 Automatic Nesting (45 minutes)

Lesson 5: Manual Nesting  
(15 minutes)

Lesson 6: Automatic Mixed Nesting  
(30 minutes)

# Lesson 5 Manual Nesting

I. FastNEST Nesting Module, desktop icon



Main functions:

1) Full Plate and Remnant Nesting -

Implement automatic cutting to improve material utilization and nesting for remnants to save material.

2) Nesting Methods -

Manual, Array, Automatic and Mixed Types.

3) High Efficiency Cutting -

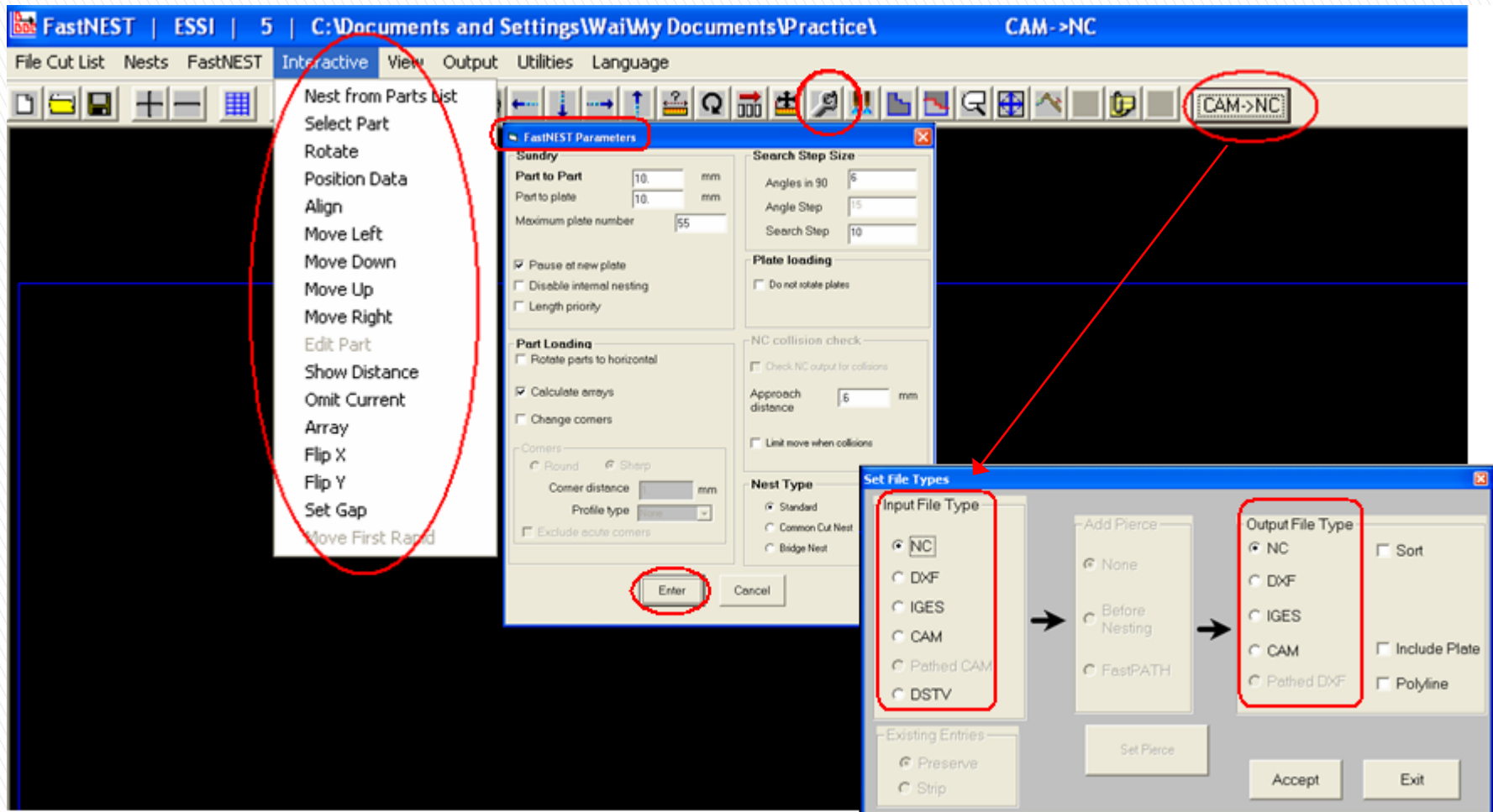
Common Edge, Bridging, Common Cutting.



# Lesson 5 Manual Nesting

## II. Manual Nesting

1. Set parameters: Input/Output file type, FastNEST parameters, Interactive nesting menu.



Manual and array nesting are the most common, straight forward and convenient methods for nesting.

Manual nesting is quite simple and obvious. Parts can be moved to appropriate areas rapidly with the left, down, right and up button. Additionally, they can be rotated with the rotation button before their movement to the desired locations.

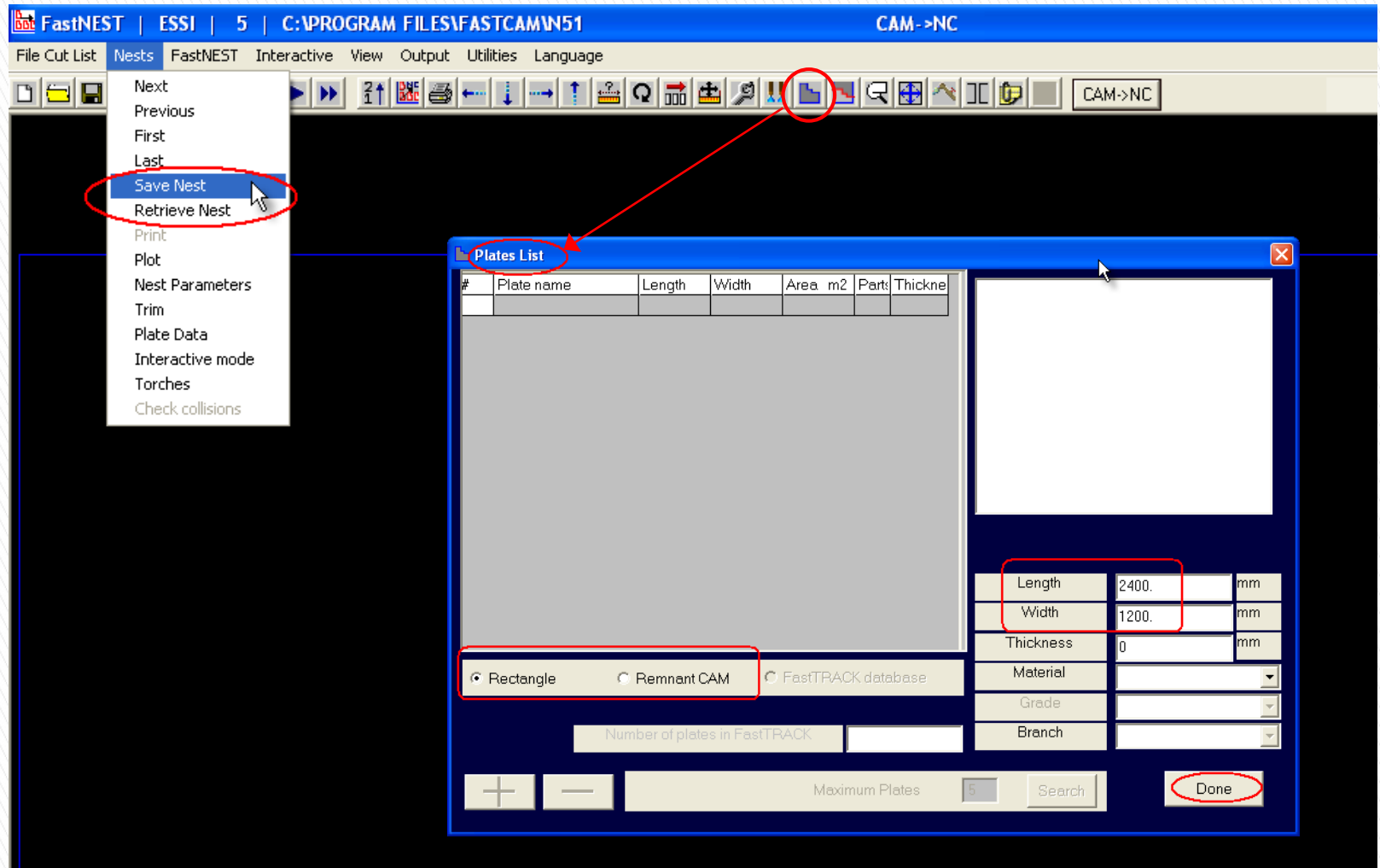
Array nesting will follow the specified rows and columns with nesting done in a quick and simple matrix style.

Regardless or whether you choose to use manual or automatic nesting, we suggest you master the basic process flow for nesting including input file type, output file format, nest parameter setting, FastPATH entry and exit.

# Lesson 5 Manual Nesting

## II. Manual Nesting

### 1. Set parameters: Plate list, Save Nest and Retrieve Nest



Setting for size of plate for nesting,  
selection of remnants, method for  
save and retrieve of nest.

# Lesson 5 Manual Nesting

## II. Manual Nesting

The screenshot shows the FastNest software interface. The main window has a menu bar with 'File', 'Cut List', 'Nests', 'FastNest', 'Interactive', 'View', 'Output', 'Utilities', and 'Language'. Below the menu bar is a toolbar with various icons. A context menu is open, showing options like 'Nest from Parts List', 'Select Part', 'Rotate', 'Position Data', 'Align', 'Move Left', 'Move Down', 'Move Up', 'Move Right', 'Edit Part', 'Show Distance', 'Omit Current', 'Array', 'Flip X', 'Flip Y', 'Set Gap', and 'Move First Rapid'. The 'Array' option is highlighted. A sub-menu for 'Automatic Array' is also visible, listing options: 'X', 'Y', 'X flipped', 'Y flipped', and 'Array in box'. The 'Cutlist' window is open, showing a table with the following data:

#	Part name	Req	Pri	Rot	Cut	Wid	Ht	Prv
1	SQUARE INCH CAM	1	5	None	1	.381	.389	0

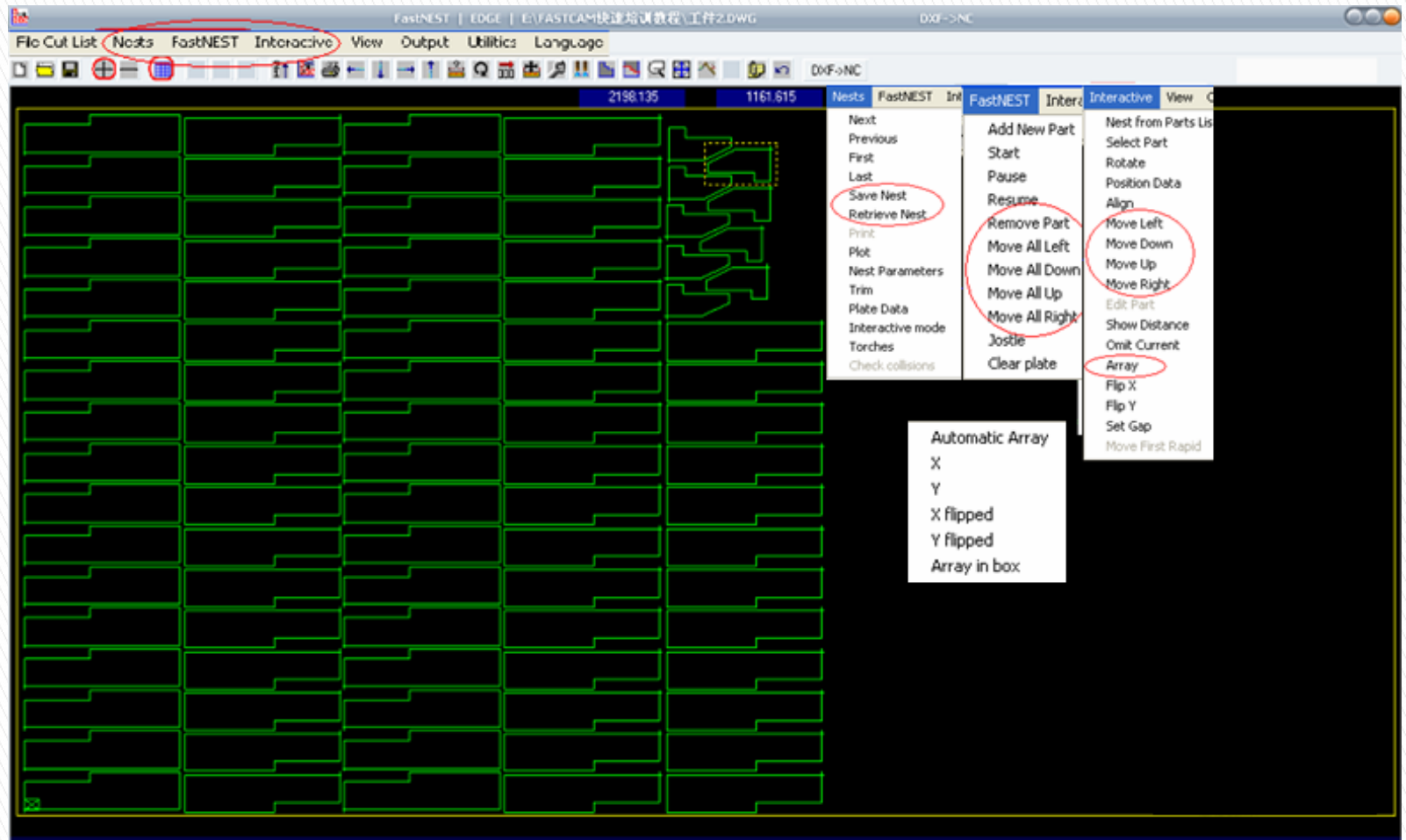
The 'Cutlist' window also features a 'Part Data' button, a 'View Nest' button, and a 'Common scale' checkbox. The main window displays a part outline with a dashed green box around it, indicating the current selection. The 'CAM->NC' button is visible in the top right corner of the main window.

## Common functions used in manual and automatic nesting:

1. Change the FastNEST parameters and plate data.
2. Movement of parts to left, down, right, up and jostle under the FastNEST menu.
3. Movement of parts to left, down, right, up, rotate and array under the Interactive menu.
4. With a part selected, double left-click will allow the change of its position (XY coordinates) and right-click can rotate part while the button is holding down.
5. While parts are being loaded on plate, right click mouse will pause the nesting process and selected parts can be deleted with the 'Delete' key on the keyboard.
6. With the parts list on screen, double left-click on part name will re-activate the dialog box for nest part data.
7. With Interactive mode under the Nest menu selected, parts will be added to plate without the display of nest part data.

# Lesson 5 Manual Nesting

## II. Manual Nesting



This screen has 80 pieces of part 1 and 9 pieces of part 2 with plate dimension of 2000 width x 3000 length:

Adding of part 1:

1. Left click on 'Add Part' button and select file name from dialog box with part 1 loaded to the plate.
2. Left click on 'Parts List' button and change required quantity in the list to 80.
3. Select 'Move All Down' under the FastNEST menu to move part to the bottom left corner.
4. With part 1 high-lighted, select 'Array' in the Interactive menu and 'X flipped' from pop up.
5. Select and delete rest of the parts with only 5 parts showing in a row displayed on the screen.
6. Pick the first part, select 'Array' under the Interactive menu followed by 'Y' from pop up to create nesting for the first column.
7. Select second part at the second column, using the same method described in last step until all 5 columns are completed.

Alternatively, array nesting can be completed automatically by direct input of the rows and columns.

Adding of part 2:

1. Left click on 'Add Part' button and select file name from dialog box with part 2 loaded to the plate.
2. Left click on 'Parts List' button and change required quantity in the list to 9.
3. Select 'Jostle' from the FastNEST menu to nest parts close together.
4. With part 2 high-lighted, select 'Array' in the Interactive menu and 'Y' from pop up to add another 4 parts in the same column.
5. Create 2nd column of part 2 with 'Array' and 'X'.
6. Rotate and move the last 4 parts to fit in the space as shown, and use 'Jostle' to keep parts close.

Final nesting completed as shown.



# Lesson 6 Automatic Mixed Nesting

## I. Automatic Nesting



Automatic nesting method:

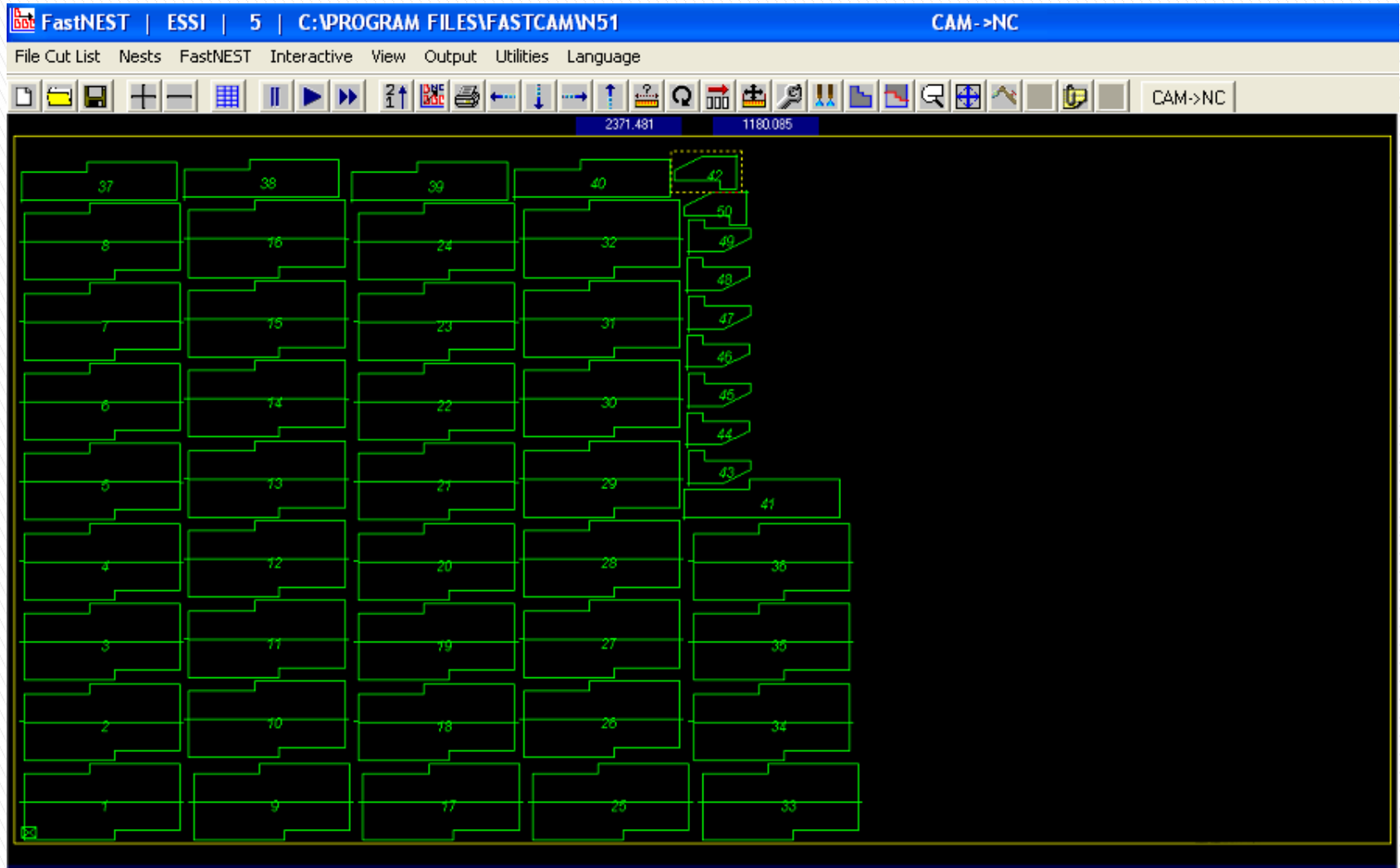
The result being shown on the screen was direct from automatic nesting without manual interruption and can be finished within seconds. In comparison with the manual nesting, the automatic option is simple, convenient and fast.

**Reminder:**

**Nesting variables must be set for automatic nesting to be more practical with better feed utilization such as rotation of parts, inputs for FastNEST parameters, nest part data ... etc.**

# Lesson 6 Automatic Mixed Nesting

## II. Automatic Mixed Nesting



**Automatic mixed nesting** is a great way to combine technology with the knowledge, skill and experience of the operator.

In automatic nesting, parts are nested according to the pre-defined parameters within the software.

For this reason, manual interruption is often used to improve material utilisation and the reasonable size of remnants. This is referred to as Automatic Mixed Part Nesting (i.e. manual + automatic nesting). Refer to the result shown in the slide from Automatic Mixed Nesting.

With Automatic Mixed nesting method, the process of nesting can be interrupted at any time for manual nesting by using the 'Esc' key. The user easily returns back to automatic nesting.

# Lesson 6 Automatic Mixed Nesting

## ▶ Manual Nesting Exercises

Ex. 1: Apply manual nesting followed by NC output for part '4.CAM' and study the result.

- Requirement to set the automatic FastPATH technique

Ex. 2: Apply manual nesting followed by NC output for part '3.DXF', note the cutting sequence/changes in FastPLOT and check costing data for job record.

- Requires to set the automatic FastPATH technique

## ▶ Automatic Mixed Nesting Exercises

Ex. 1: Apply automatic mixed nesting followed by NC output for part '6.DXF' and study the result.

- Requires to set the automatic FastPATH technique

Ex. 2: Apply automatic mixed nesting followed by NC output for part '5.DXF', observe cutting sequence/changes in FastPLOT. Check costing data and compare with the result from Ex. 2 of manual nesting to understand the benefits from automatic mixed nesting ...

- Requires to set the automatic FastPATH technique (must be with same setting as Ex. 2 of the manual nesting)

5 min. break

## Module 4 Optimized Cutting (45 minutes)

Lesson 7: Automatic Common Cutting (15 minutes)

Lesson 8: Plasma Bridging (15 minutes)

Lesson 9: Automatic Continuous Cutting  
(15 minutes)

Lesson 10: Nesting exercises (15 minutes)

NC cutting efficiency is mainly affected by:

1. Hole piercing in Oxy flame cutting.
2. Piercing and replacement of the electrode in Plasma cutting.

The intention of high efficiency cutting is to reduce the piercing of holes in Oxy and Plasma cutting. This offers a productivity improvement and directly helps save consumables & parts.

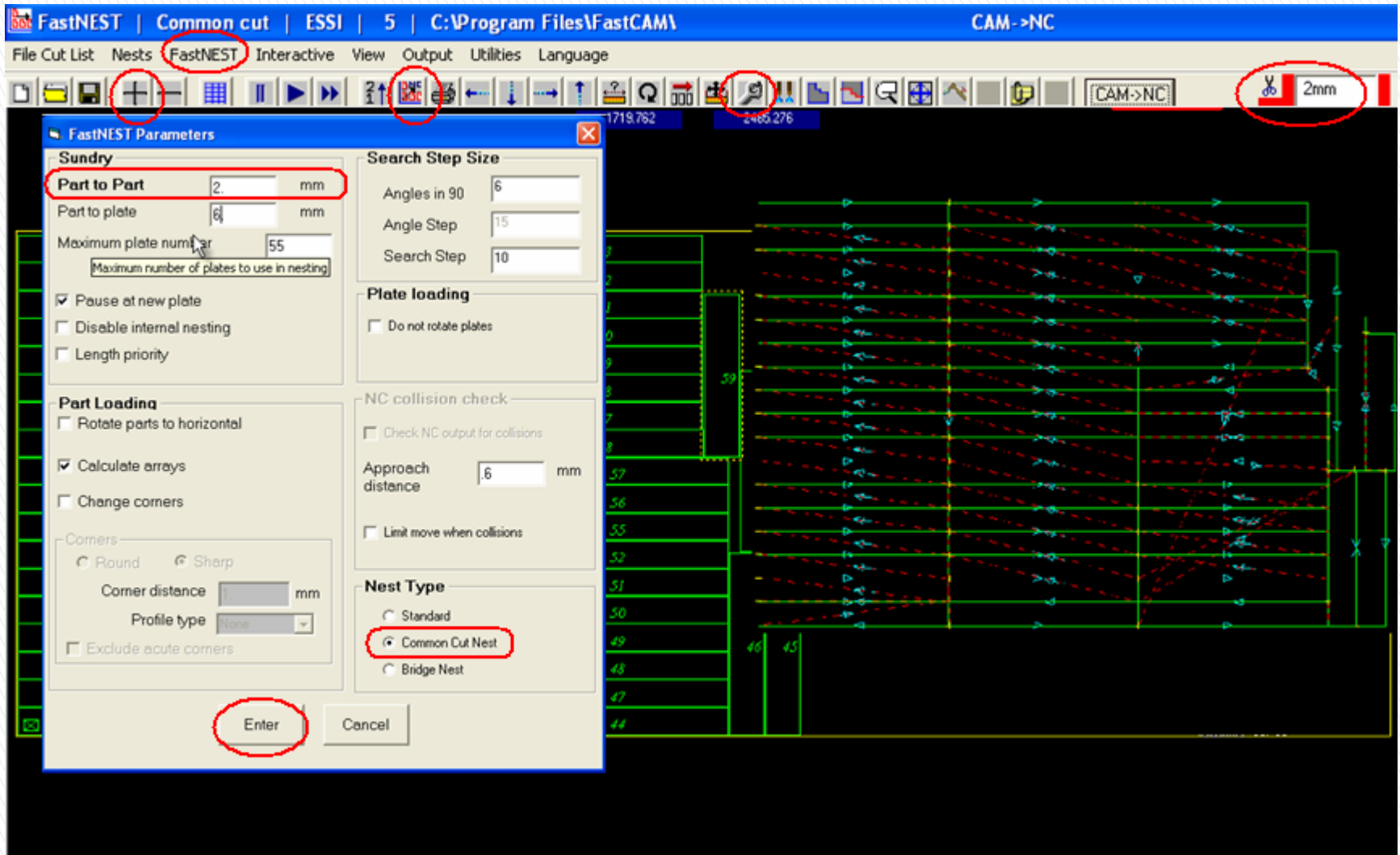
Optimized Cutting options:

1. Common cutting – typically applies to oxy, plasma, LASER and water jet cutting. With common cut nesting, 30% of cutting paths can be saved and the amount of pre-heat and pierce holes can be reduced.
2. Bridge cutting – mainly used in plasma cutting for continuous cutting to reduce hole piercing and cost of consumables.
3. Continuous cutting – used in oxy, plasma, LASER and water jet cutting. Reduces pre-heat and hole piercing and offers productivity improvements by connecting entry and exit of parts in nesting for continuous cutting.



# Lesson 7 Automatic Common Cutting

## I. Automatic common edge cutting



Application steps for common cutting:

1. Open the FastNEST software.
2. Click on the 'Change Parameters' button, set part to part distance to 1 time of the kerf value, select common cut for Nest Type and enter.
3. Enter/Change size of the plate, add and nest parts on plate.
4. NC output.
5. Check and verify start point and cut path of common edge parts before start of cutting on machine.

Note:

1. Common cut nest type must be selected before the addition of parts on plate for this function to work.
2. The effect of thermal distortion caused from location of entry and direction of cutting paths.
3. Except the use of normal pre-heat time for the first pierce hole, others have to follow the edge pre-heat guideline and start cut within 1 minute.

Reminder:

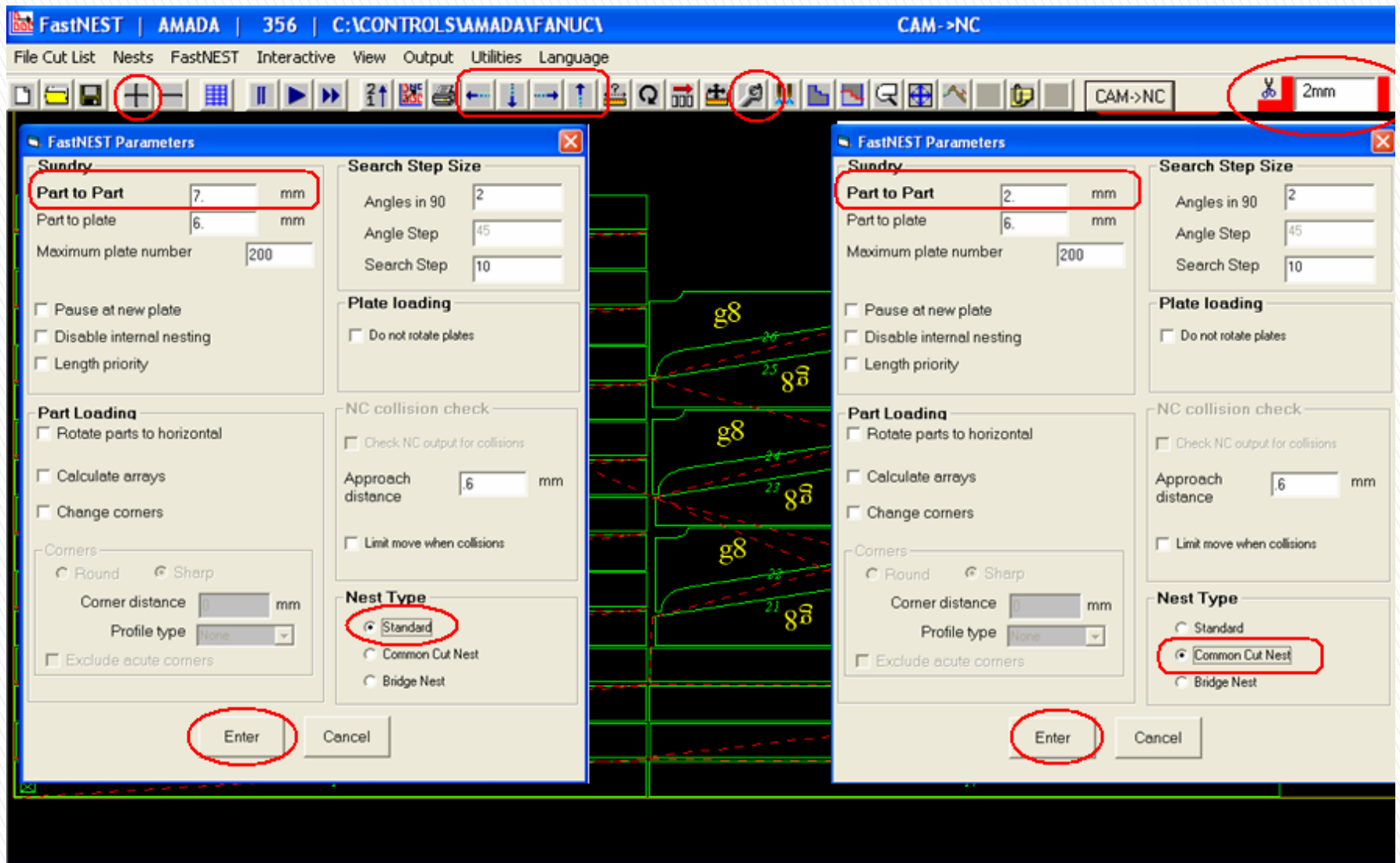
For automatic common cutting, kerf will be compensated by the software according to part to part value specified in the FastNEST parameters (i.e. 1 time of the kerf value in diameter).

## **Common Cut Pairs**

Common cut pairs are defined by joining two identical parts along a common straight boundary. This can reduce the cutting and halve the piercing. It is used most often in shipbuilding as it is common to encounter left and right hand pairs.

# Lesson 7 Automatic Common Cutting

## II. Mixed Automatic Common Edge Cutting



## Mixed applications (with kerf compensation in output)

1. Start FastNEST from icon or the start menu.
2. To minimize gaps between parts, clear selection for 'Existing Entries' shown at bottom left corner of the Set File Types dialog box.
3. Open the nest parameters box, activate common cut for nest type, change 'Part to Part' distance to common edge value (i.e. 1 time of the kerf value) and click enter with the common cut value displayed at right hand side of the toolbar.
4. Add and nest required parts for common edge cutting (note they have to be arranged together) then use the 'Jostle' function to adjust parts close to each others (Note: When using automatic nesting, it is necessary to check if position and start point of nesting are the same or it may not be possible for common cut to occur).
5. Re-enter the 'Part to part' value for standard nest type (i.e. must be greater than 2X kerf value) and change the Nest Type back to Standard.
6. Add remaining parts which do not require common edge cutting and proceed nesting.
7. Back to the nest parameters box, set 'Part to part' distance back to the common cut value and re-activate common cut for nest type.
8. Output NC (this must be done while common cut nest type is active).
9. Verify, check and confirm there is no error in cutting positions and paths before NC is sent to machine for cutting.

### Note:

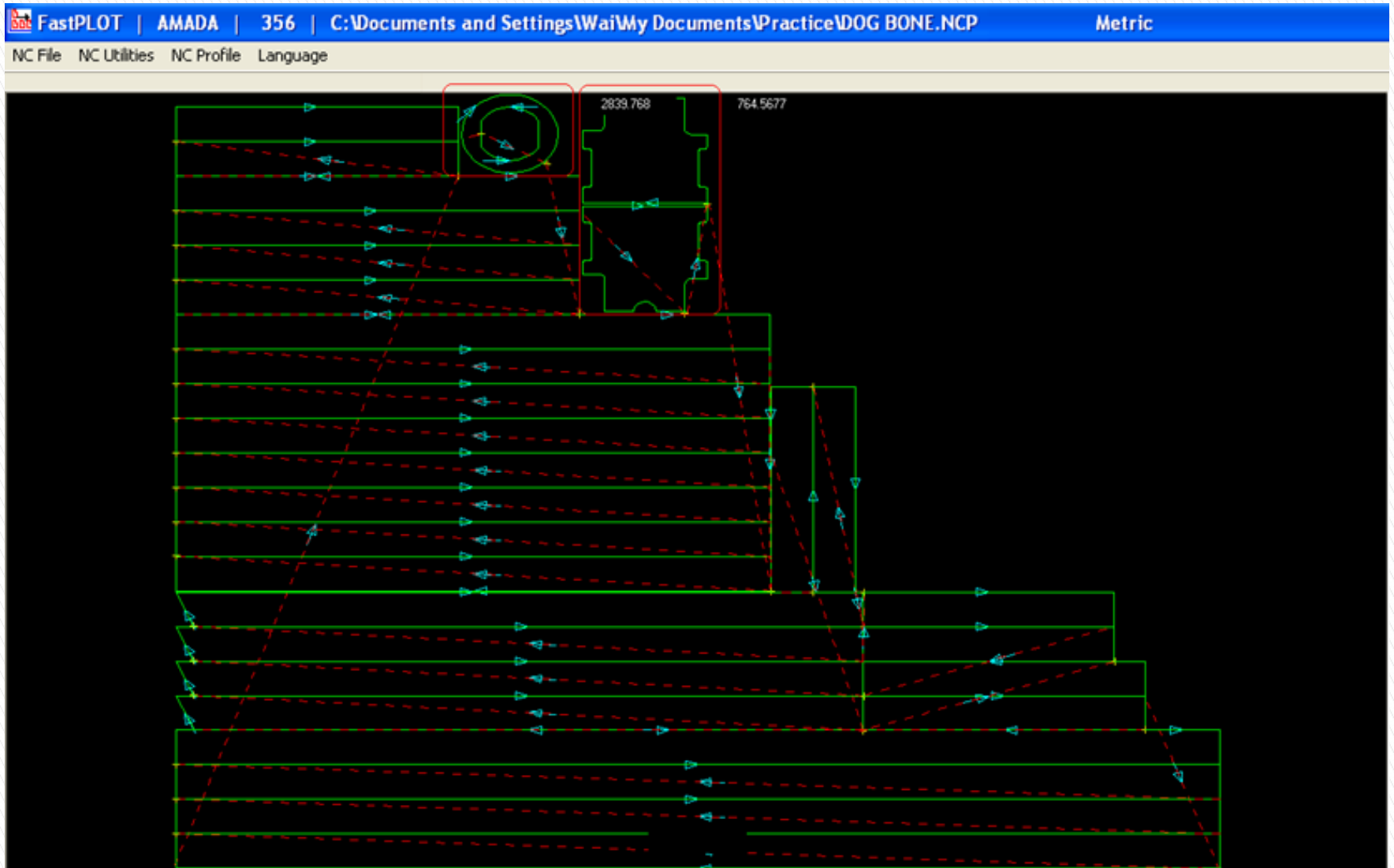
1. Function of re-sequence cannot be reused in above steps.
2. Automatic nesting will not be affected after the last activation of common cut nest just before the NC output.
3. Take note of following when using the common cut function.
  - i) The effect of thermal distortion from position of entries and direction of the programmed paths.
  - ii) Except the use of normal pre-heat time for first hole of common cut and rest of holes for standard cut, edge pre-heat should be used for other part of common cut nests i.e. max. 1 minute before start of cutting.

### Reminder:

For automatic common cutting, kerf will be compensated by software according to part to part value specified in the FastNEST parameters (i.e. 1 time of the kerf value in diameter).

# Lesson 7 Automatic Common Cutting

## II. Mixed Automatic Common Edge Cutting



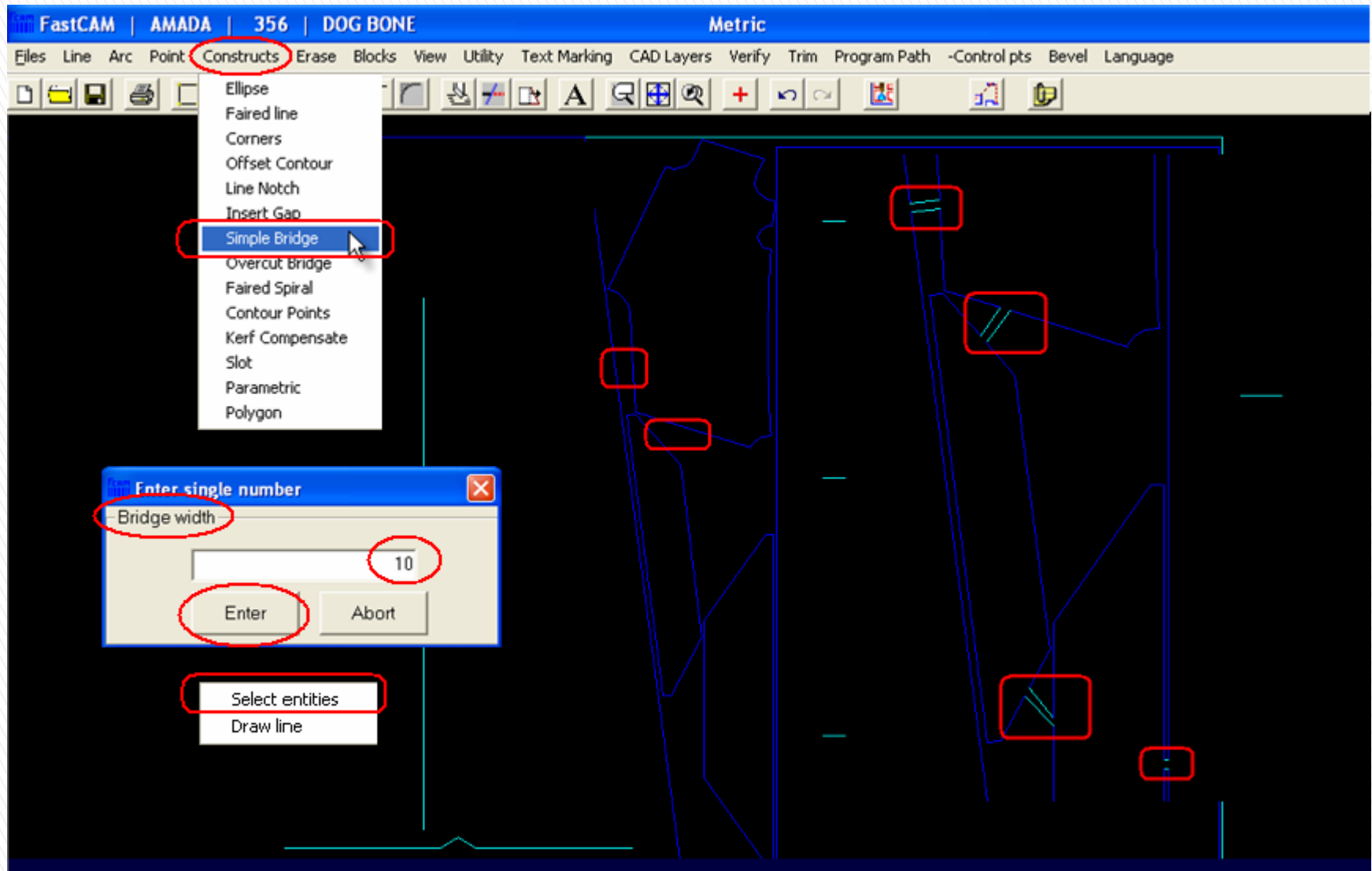
As illustrated, the common cut nest process has not been used for parts enclosed in red rectangles.

### Description:

To prevent incorrect piercing of holes on parts due to 'Part to part' distance set in FastNEST is less than length of entries and exits, a logical adjustment has been added to reduce lengths of entries and exits same as the 'Part to part' distance in new versions.

# Lesson 8 Bridges for Plasma Cutting

## I. FastCAM Bridges





The Bridge function for plasma cutting is used to link parts into groups with bridges such that the linked parts can be cut continuously with only one pierce hole. This function aims at reduction of the pierced holes for lower consumable costs and higher productivity plus minimization of heat distortion.

Steps in using bridging in FastCAM:

1. Change output file type in FastNEST to the CAM type.
2. Use of automatic mixed nesting and output as CAM file.
3. Open the CAM file in FastCAM, select 'Constructs' → 'Overcut Bridge' or 'Simple Bridge' → set bridge width → Enter.
4. Select outer contour of entities to be linked with bridge and form a new external contour.

Beware:

1. Travel direction of entry and external contour in relation to thermal distortion.
2. Position of bridges and travel direction of contours in relation to thermal distortion.
3. Beware of potential problems from newly created internal contours and ensure they will be cut first according to the principle.
4. Beware of the formation of dead loops.

# Bridging Information Update for v5.10

FOUR different types that will be included in the Professional Edition. Bridging can be combined with other processes such as line marking and text marking.

- ▶ 1) **(Traditional) Bridging.** A simple positive bridge is where parts are connected together linked by two parallel cuts (bridge). Bridges allow continuous cutting and are traditional in many shipbuilding applications. The result is much faster cutting. Parts are left joined, but this can be an advantage in some circumstances. These include making assemblies, moving the parts as a group and minimizing part movement (appropriate for lighter materials or thinner materials).
- ▶ 2) **Overcut or Negative Bridging** allows for continuous cutting with the immediate separation of parts. This is a FastCAM® invention that has the advantages of bridging but cuts out the bridge on the return. Previously available in the FastCAM® drawing editor, this function is now available in FastNEST® where it is perhaps more intuitive. Selecting this function will deliver a speed increase on many nests and result in a reduction in piercing making it a feature of interest to a wider range of customers.
- ▶ 3) **Skip Bridging.** Here you have additional piercing and the parts are held together by a bridge but the bridge is not cut. This can be used for routing aluminum, as it prevents vibration but does involve piercing on the edge, which is fine for routing. You actually do much more piercing and save nothing on cutting, so it is a device mainly to prevent vibration and movement. A simple gap bridge forms much the same function for lighter materials where the resultant tabs can be broken manually. In this case, additional pierces are required rather than fewer. (This function is aimed at aluminium routers but can be used in any instance where the programmer does not want the bridge to be cut).
- ▶ 4) **Skip with Entries.** As with Skip Bridging, but pierces are away from the part edge.

# Lesson 8 Bridges for Plasma Cutting

## II. FastNEST Bridges

The screenshot displays the FastNEST software interface for plasma cutting. The main window shows a list of files (File Cut List, Nests, FastNEST, Interactive, View, Output, Utilities, Language) and a toolbar with various icons. The **CAM->NC** button is highlighted with a red circle. The **10mm** dimension is also highlighted with a red circle.

The **FastNEST Parameters** dialog box is open, showing the following settings:

- Sundry**
  - Part to Part: 2. mm
  - Part to plate: 6. mm
  - Maximum plate number: 10
  - ☐ Pause at new plate
  - ☐ Disable internal nesting
  - ☐ Length priority
- Part Loading**
  - ☐ Rotate parts to horizontal
  - ☒ Calculate arrays
  - ☐ Change corners
- Corners**
  - ☐ Round
  - ☒ Sharp
  - Corner distance: 0 mm
  - Profile type: None
  - ☐ Exclude acute corners
- Search Step Size**
  - Angles in 90: 12
  - Angle Step: 7.5
  - Search Step: 10
- Plate loading**
  - ☐ Do not rotate plates
- NC collision check**
  - ☐ Check NC output for collisions
  - Approach distance: .6 mm
  - ☐ Limit move when collisions
- Nest Type**
  - ☐ Standard
  - ☐ Common Cut Nest
  - ☒ Bridge Nest

The **Enter** button is highlighted with a red circle.

The **FastPATH Settings** dialog box is also open, showing the following settings:

- Router**
  - Depth on each pass: 0 mm
- Bridges**
  - Bridge Type: Bridge
  - Bridge Width: 10. mm
  - Entry Length: mm

The **Bridge** button is highlighted with a red circle. The **Bridge Width** field is also highlighted with a red circle.

A central menu is visible with the following options:

- Add
- Delete
- Delete last
- Delete all
- Show bridged edges
- Set parameters

The background shows a plasma cutting path on a plate, with red boxes highlighting specific areas of the path.

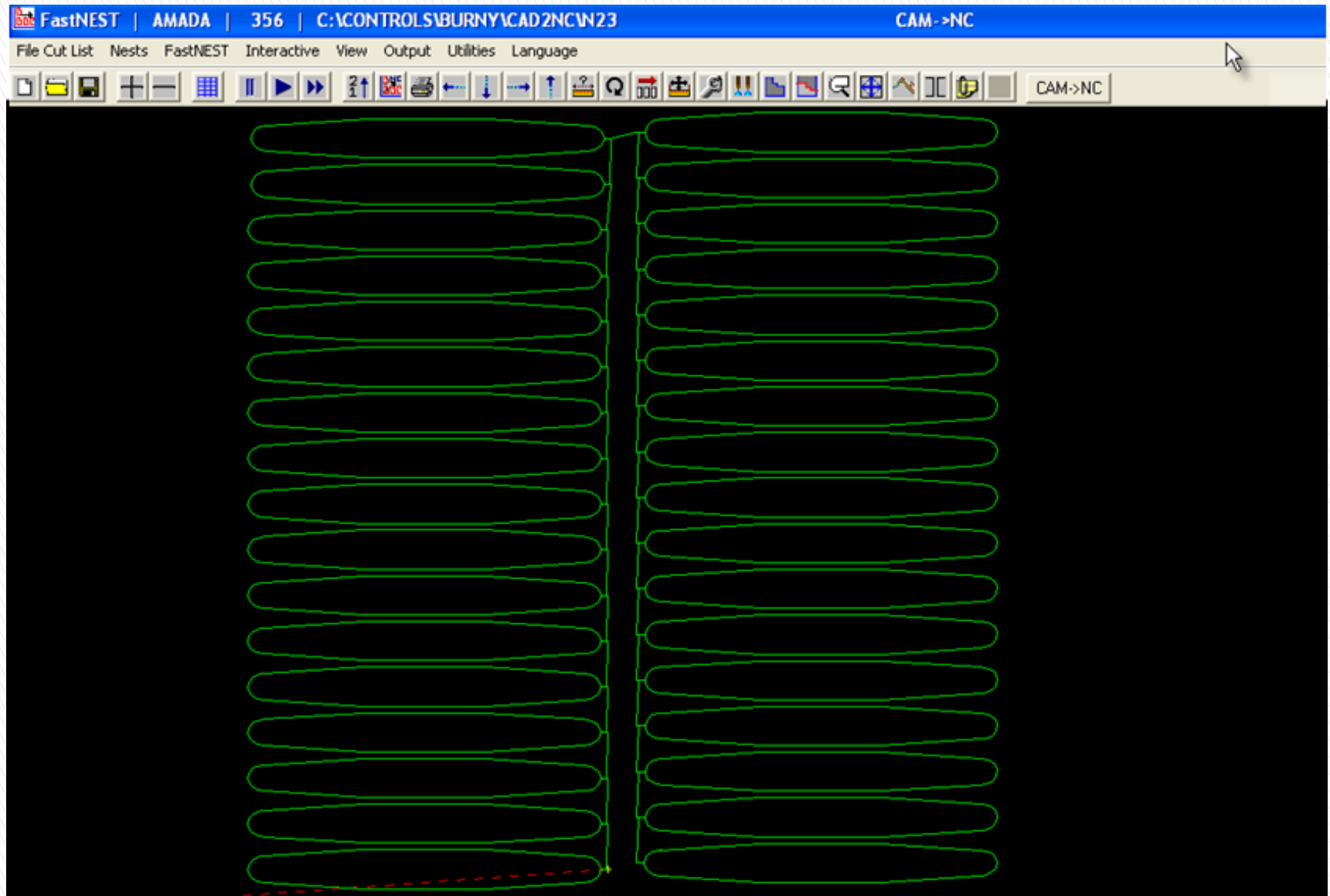
## Steps:

1. Select 'Add Pierce' before nesting in the 'Set file types' box, followed by 'Set Pierce' to set the position of entry in FastPATH.
2. Select 'Bridge Nest' for Nest Type in the 'Nest Parameters' box and enter width of bridge in the pop up if it is the first time to use this function. Then the amount of bridge width will be displayed on right hand side of the toolbar with the 'Add Bridge' button activated between the Gas Axe and Email button.
3. Add and nest parts as normal, click on the 'Add Bridge' button and select Add, Delete, ... from the pop up.
4. Pick outer contours of parts needed to be linked, build a bridge between 2 contours to form a new external contour.

## Beware :

1. Travel direction of entry and external contour in relation to thermal distortion.
2. Position of bridges and travel direction of contours in relation to thermal distortion.
3. Beware of potential problems from newly created internal contours and ensure they will be cut first according to the principle.
4. Beware of the formation of dead loops.

# Lesson 9 Automatic Continuous Cutting



## Steps:

1. Adding of parts as normal with FastNEST.
2. To facilitate continuous cutting, you must consider the position of entry/exit and adjust their position as required (Position of entry/exit can also be changed during the NC output process).
3. Select 'Cut Short Rapids' under Nest Options of the FastPATH settings and enter the shortest distance between exit of last part and entry of next part – distance of Short Cut value.
4. NC output and plot for verification.

Pay attention to the following before the use of this function:

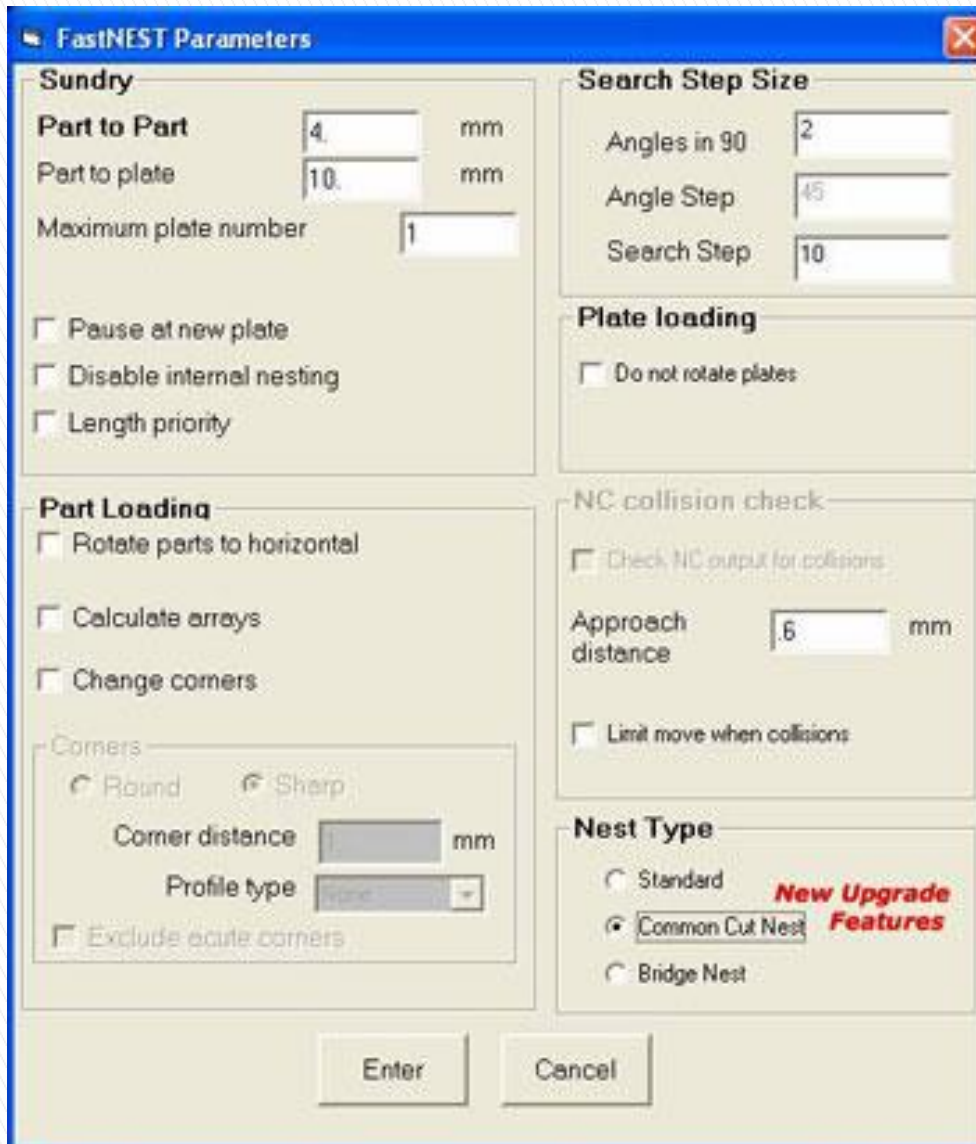
1. Travel direction of entry and exit in relation to thermal distortion.
2. Travel between the entry and exit will not pass over contour of parts.
3. Refer to demonstration for practical steps.

## Lesson 10 Nesting Exercises

1. With common cutting, prepare nesting for part '6.DXF' and '7.DXF' then generate the NC outputs. Using the same parameters, check in FastPLOT and record the costing data.  
Requirement : Kerf 2 mm, Plate size 6,000 mm and 4,000 mm, Part quantity 30 and 18.
2. With mixed common cutting, prepare nesting for part '6.DXF' and '7.DXF' then generate the NC outputs. Using the same parameters, check costing data in FastPLOT and compare result from Ex. 1.  
Requirement : Kerf 2 mm, Plate size 6,000 mm and 4,000 mm, Part quantity 30 and 18.
3. With bridge cutting turned on in FastNEST, prepare nesting for part '6.DXF' and '7.DXF' then generate the NC outputs. Using the same parameters, check costing data in FastPLOT and compare result from Ex. 1.  
Requirement : Kerf 2 mm, Plate size 6,000 mm and 4,000 mm, Part quantity 30 and 18.



# Common Cut Information Update



The image shows a screenshot of the 'FastNest Parameters' dialog box. It is divided into several sections: 'Sundry', 'Search Step Size', 'Plate loading', 'Part Loading', 'NC collision check', 'Corners', and 'Nest Type'. The 'Sundry' section includes input fields for 'Part to Part' (4 mm), 'Part to plate' (10 mm), and 'Maximum plate number' (1), along with checkboxes for 'Pause at new plate', 'Disable internal nesting', and 'Length priority'. The 'Search Step Size' section has input fields for 'Angles in 90' (2), 'Angle Step' (45), and 'Search Step' (10). The 'Plate loading' section has a checkbox for 'Do not rotate plates'. The 'Part Loading' section has checkboxes for 'Rotate parts to horizontal', 'Calculate arrays', and 'Change corners'. The 'Corners' section has radio buttons for 'Round' and 'Sharp' (selected), a 'Corner distance' input field, a 'Profile type' dropdown menu, and a checkbox for 'Exclude acute corners'. The 'NC collision check' section has a checkbox for 'Check NC output for collisions', an 'Approach distance' input field (6 mm), and a checkbox for 'Limit move when collisions'. The 'Nest Type' section has radio buttons for 'Standard', 'Common Cut Nest' (selected), and 'Bridge Nest'. A red text label 'New Upgrade Features' is placed next to the 'Common Cut Nest' option. At the bottom are 'Enter' and 'Cancel' buttons.

**FastNest Parameters**

**Sundry**

Part to Part: 4 mm  
Part to plate: 10 mm  
Maximum plate number: 1

☐ Pause at new plate  
☐ Disable internal nesting  
☐ Length priority

**Search Step Size**

Angles in 90: 2  
Angle Step: 45  
Search Step: 10

**Plate loading**

☐ Do not rotate plates

**Part Loading**

☐ Rotate parts to horizontal  
☐ Calculate arrays  
☐ Change corners

**Corners**

☒ Round ☒ Sharp  
Corner distance: mm  
Profile type: mm  
☐ Exclude acute corners

**NC collision check**

☐ Check NC output for collisions  
Approach distance: 6 mm  
☐ Limit move when collisions

**Nest Type**

☐ Standard ☒ Common Cut Nest ☐ Bridge Nest

**New Upgrade Features**

Enter Cancel

In FastCAM® Professional v5.10, **we have introduced the ability to completely common cut nest.** Compared with traditional nesting of separate parts each with their own external entry and exit, FastCAM® Common Cut Nest gives you the ability to set the part spacing to exactly the cut width before nesting. See next slide.



Compared with traditional nesting of separate parts each with their own external entry and exit, FastCAM® Common Cut Nest gives you the ability to set the part spacing to exactly the cut width before nesting.

**FastCAM® v5.10 achieves 'zero gap' common cutting** by cutting entire nests using a start on (part) edge approach. In many nests this will usually eliminate piercing time and halve cutting time. Intelligent strong/weak analysis logic is built into the software to minimize part movement however there is the potential for slightly damaged parts so part quality is a consideration. There is also the potential of damage to the part if an edge start command is not present in the NC control. However it offers the possibility of many common cuts in a nest of largely rectilinear parts where quality is not the main concern. It is dramatically faster with far less piercing. For rectilinear parts such as nests of rectangles, the saving in piercing can be 5x. The saving in cutting can be 2x, yielding gains of 10x in total processing speed!

Like any new technology, there are advantages and disadvantages depending on the application. The NC programmer can quickly weigh up the pro's and con's on a job by job basis. Certainly if you're cutting barbeque plates or similar then the efficiencies using the new Common Cut Nest feature can be remarkable.

## Optimized Cutting in Review:

FastCAM offers optimized cutting techniques and continues to develop new methods for:

1. Common cutting,
2. Bridge cutting, and
3. Continuous cutting.

Use of these functions yields improvements such as:

1. Reduction of pierce holes effectively by 70%,
2. Reduction of cutting paths by 30%,
3. Improvement of cutting efficiency by 40%, and
4. Reduction of consumable items such as the plasma torches, electrodes and LASER lens by 40%.

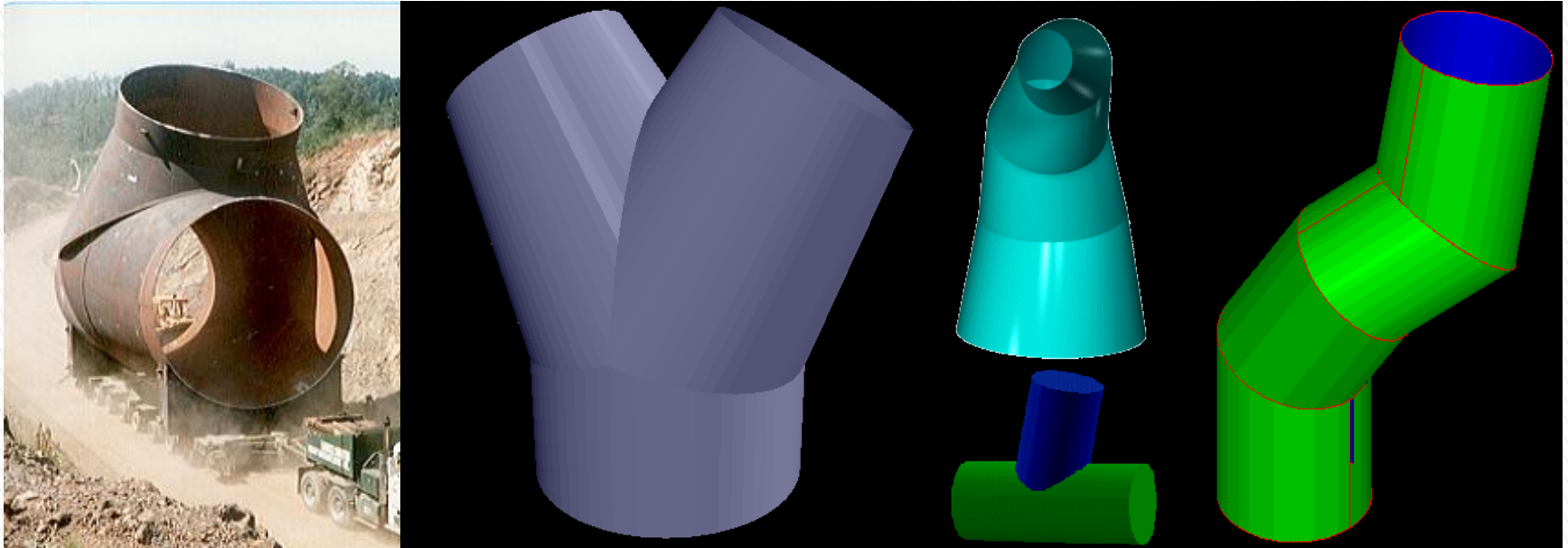
5 min. break

# Module 5 FastCAM Solutions

**FastCAM software isn't limited to drawing and nesting! One of the benefits of having the largest range of software for steel fabrication means that over the years, we've solved many unique and complex fabrication problems. Our software is often a central aspect of the total engineering solution.**

**We highlight a few popular programs that can be used in conjunction with FastCAM or purchased as stand alone programs.**

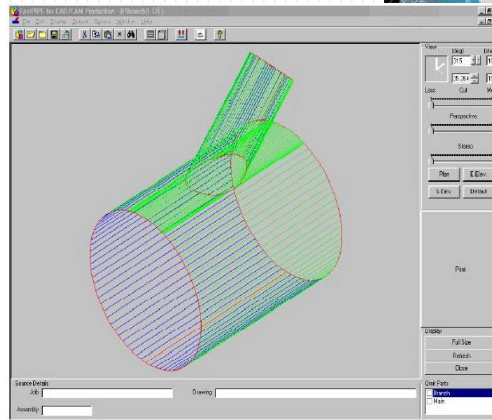
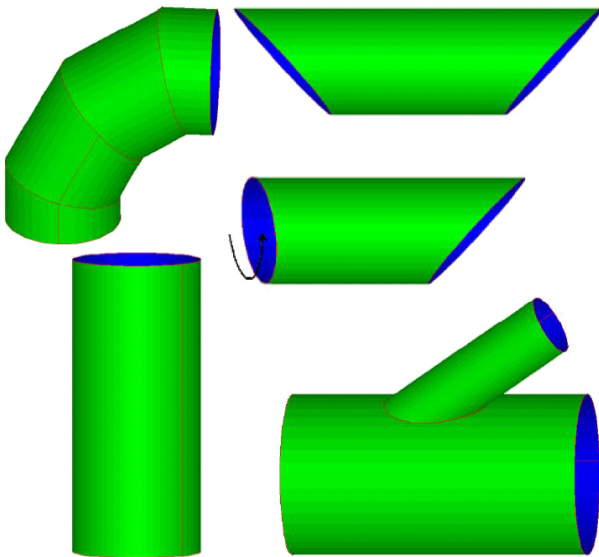
# I. FastSHAPES® Heavy Plate Development



Automate fabrication/weld calculation and layout of multi segment heavy plate transitions, boilers, blowpipe, power, transitions. Huge saving of skilled manpower required to calculate unfolding and fitup.

## II. FastPIPE™

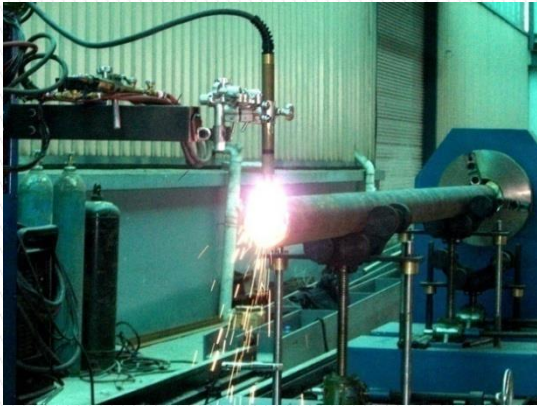
### The Fast Way to Profile Pipe Ends for Pipe to Pipe & Reticulation



FastPIPE simplifies & speeds All Types of  
Pipe Development & Estimation work  
Produces Wrap around templates with  
full Weld Preparation details

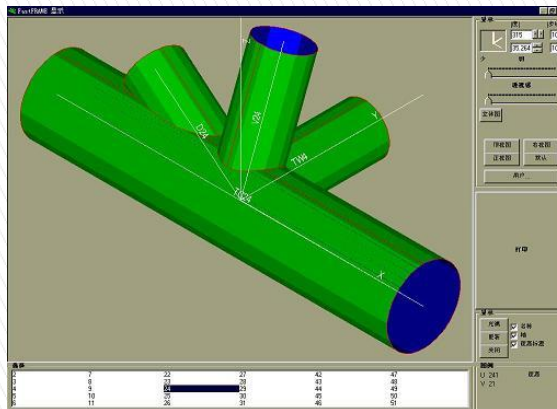


### III. FastFRAME® Complex Tubular Space Frames



Saves hundreds of hours in the design stage of complex space frame developments. Complete structural frames can be completed in days!

**Of equal value, the extraction of fabrication data needed for processing is simple!**



## IV. FastCAM® QE™

### Optimized Estimates for Processed Steel

For Burning, Shearing, Sawing, Drilling

Comprehensive, Purpose Built  
Quotation System for Metal  
Service Centers.

Contains both Linear and Plate  
based optimization with the  
least percentage scrap.



Produces an Email or fax ready quote after nesting, with full allowance for all remnant, scrap and processing costs.

Profiling, Shearing: Parametric, True Shape, Rectangular  
Sawing, Drilling for Long products, Beams, Channels, Angles etc.



## V. FastCOPY® for Parts & Drawings

Offline system for fast, accurate conversion of existing parts, templates, or drawings into DXF & NC code ready for 'best quality' cutting. Easier than an electronic eye. Includes a new design lightweight portable 'plug and play' digitizer for ease of use and storage. Packs away in minutes.



Adds value to any busy service center or business needing to copy and profile cut complex parts or precision artwork from metal. Destroy/archive old paper drawings and instantly retrieve templates. Easy to use.

## VI. FastCUT® Optimizer

- ▶ **Optimized layouts for general cutting of lengths or rectangles**, shears and saws (plate & bar). Nests then prints graphic cutting lists. DXF output. Use for basic costing information.



Saves material and operator time.  
Improves time to quote. Reduces cutting errors and layout time.

## VI. FastTRACK® Plate Tracking

### Audit Trail

Audit Id	Entry Number	Filename	Description	Date	Time	User	Computer Name
8	2	GR201.CAM	Plate Deleted.	30/08/2006	03:30:39 PM	TREVOR	TREVOR
7	1	GR201	Plate Deleted.	30/08/2006	03:30:35 PM	TREVOR	TREVOR
6	2	GR201.CAM	Plate shape has been edited.	30/08/2006	03:28:24 PM	TREVOR	TREVOR
5	2	GR201.CAM	Plate shape sent for editing.	30/08/2006	03:28:24 PM	TREVOR	TREVOR
4	2	GR201.CAM	New CAM file added.	30/08/2006	03:27:41 PM	TREVOR	TREVOR
3	1	GR201	Plate shape has been edited.	30/08/2006	03:24:40 PM	TREVOR	TREVOR
2	1	GR201	Plate shape sent for editing.	30/08/2006	03:24:40 PM	TREVOR	TREVOR
1	1	GR201	New Rectangle added.	24/08/2006	12:52:47 PM	TREVOR	TREVOR

### Plate Selection

Material and Grade:

☒ Thickness:  mm →  mm

☒ Only Active Plates

### Plate Information

Original Size:  X  mm

Current Size:  X  mm

Remaining:  % ☐ Show Text ☒ Show original plate

Cost:

### FastTRACK Spreadsheet

Entry number	Plate name	Material	Grade	Thickness mm	Area m2	Density kg/m3	Weight kg	Location	Heat Number	Mf Number	Order	Is
5	GR20005	Mild Steel	GR250	25	19.2	7850	1508					
6	GR20006	Mild Steel	GR250	25	21.6	7850	1700					
7	GR20007	Mild Steel	GR250	12	14.8	7850	1160					
8	GR20008	Mild Steel	GR250	6	2.09	7850	163					

Beginning Previous Next End

Select Rules:

Choose Material:  Choose Grade:

Choose Thickness:  ☐ Only Active Plates

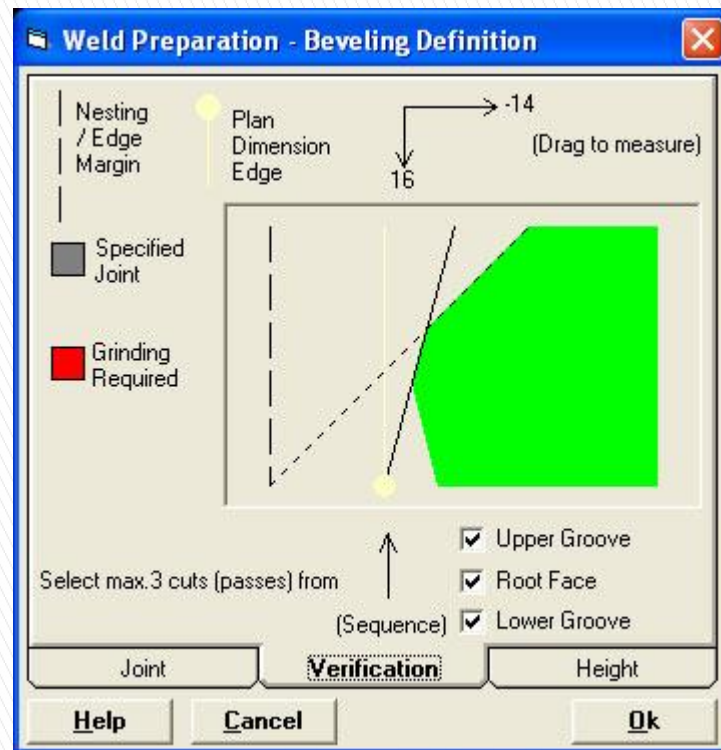
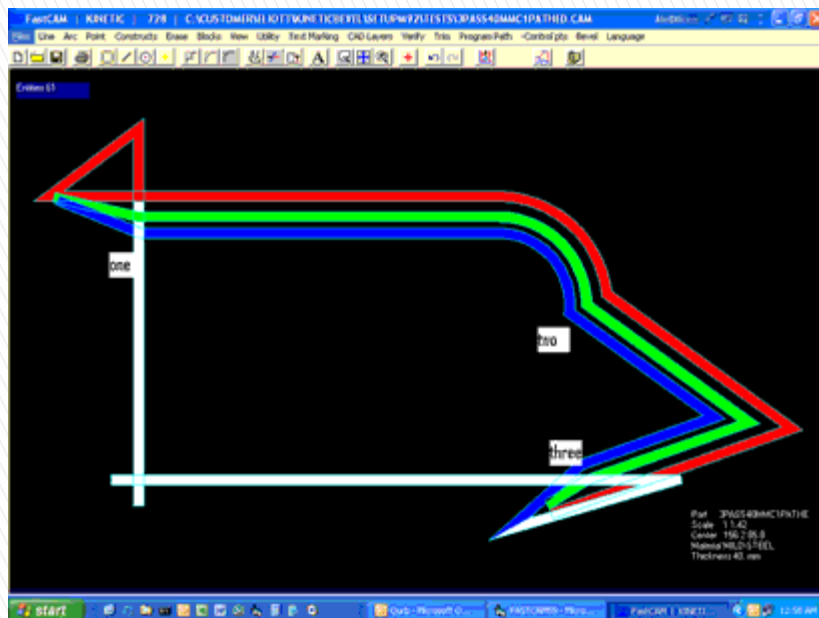
Choose Heat Number:

Choose Location:

Maximum number of blocks at: 1

Minimizes stock holding of partially processed plates and uses the most cost effective plate stock for the job. Fast stock take & auditing system.

## VII. FastCAM Bevel for Weld Ready Parts



Ultra simple automatic 3-Pass beveling to achieve weld ready parts (true K,Y engineering parts for heavy plate forming).

## **Some features that you can add to your FastCAM® System**

### **include:**

- ▶ Text marker is useful plate marking software for NC machines fitted with Scribes, Punch Markers, Plasma Etch markers & Ink-Jet. Converts text & numbers into stroke font movements. Uses stroked characters for minimal machine movement. Ideal for lasers and/or plasma with low power torches. Requires FastCAM®.
- ▶ FontGEN converts 'Windows' fonts into a format for NC cutting. Ideal for cutting all styles of letters and fonts such as used for signwriting etc.
- ▶ Outline converts pictures or shapes ready for cutting. Ideal for converting non-dimensioned artwork into cutting files. Ornamental iron work, signs, machinery logos etc. Reads most graphic formats.
- ▶ NC Collision Checker replaces the visual checks in both FastNEST and FastPLOT to be computer based and automatic. Of value in big and complex nests where a small overlap can easily be missed resulting in expensive cutting mistakes.
- ▶ FastLINK provides a fast DNC file download link from older controllers which speeds up processing.

**Did you know that FastCAM® Profiling Systems support multiple posts?**

If you replace or add another type of cutting machine you don't need to change your programming system!

If you are on an Annual Service Agreement, we will support you getting your machines up and running! If you are not on a Service Contract we will provide technical support at current rates. We invite you to call us to discuss machine specific details.

Note: Excludes FastCAM® Systems marked as 'OEM' versions (Specific machine bundled software).



**FastCAM programs that use a physical hardware security lock or dongle are:**

**FastCAM FastPLOT FastNEST FastSHAPES FastTRACK  
FastCOPY FastLINK**

- ▶ **What happens if you make changes to your PC and you use one of these 'dongles'?**
- ▶ Changes to your network card or any other internal hardware, upgrading your operating system etc should make no difference. If you buy a complete new PC then just make sure that you remove the dongle from the old PC and attach it to the new one.
- ▶ If you require a USB dongle you only need to purchase a new holder (outer case) from us. We will also send a simple instruction sheet on how to transfer your unique ID 'button' from one holder to another.

# Conclusion

Today FastCAM has many more products encompassing many industries and distribution points and FastCAM is used in many different environments around the world.

- **STEEL SERVICE CENTERS**
- **STEEL FABRICATION**
- **METAL FABRICATION**
- **MINING**
- **CONSTRUCTION**
- **SIGNCUTTING**
- **WOODWORKING**
- **FLAT GLASS**
- **EDUCATION**



# End of Tutorial Thank You!

Please visit [www.fastcam.com](http://www.fastcam.com)  
for more information.