Mastercam X3

# TEST PROJECT

# TEACHER GUIDELINES

# Objectives

You will create the geometry for Pen Base & Acylic Plaque, and then generate a toolpath to machine the part on a CNC machine. This projecct covers the following topics:


➲ **Create a 2-dimensional drawing by:**
Creating a Rectangle
Creating Arcs
Creating Lines
Creating Mastercam geometry by converting an image file

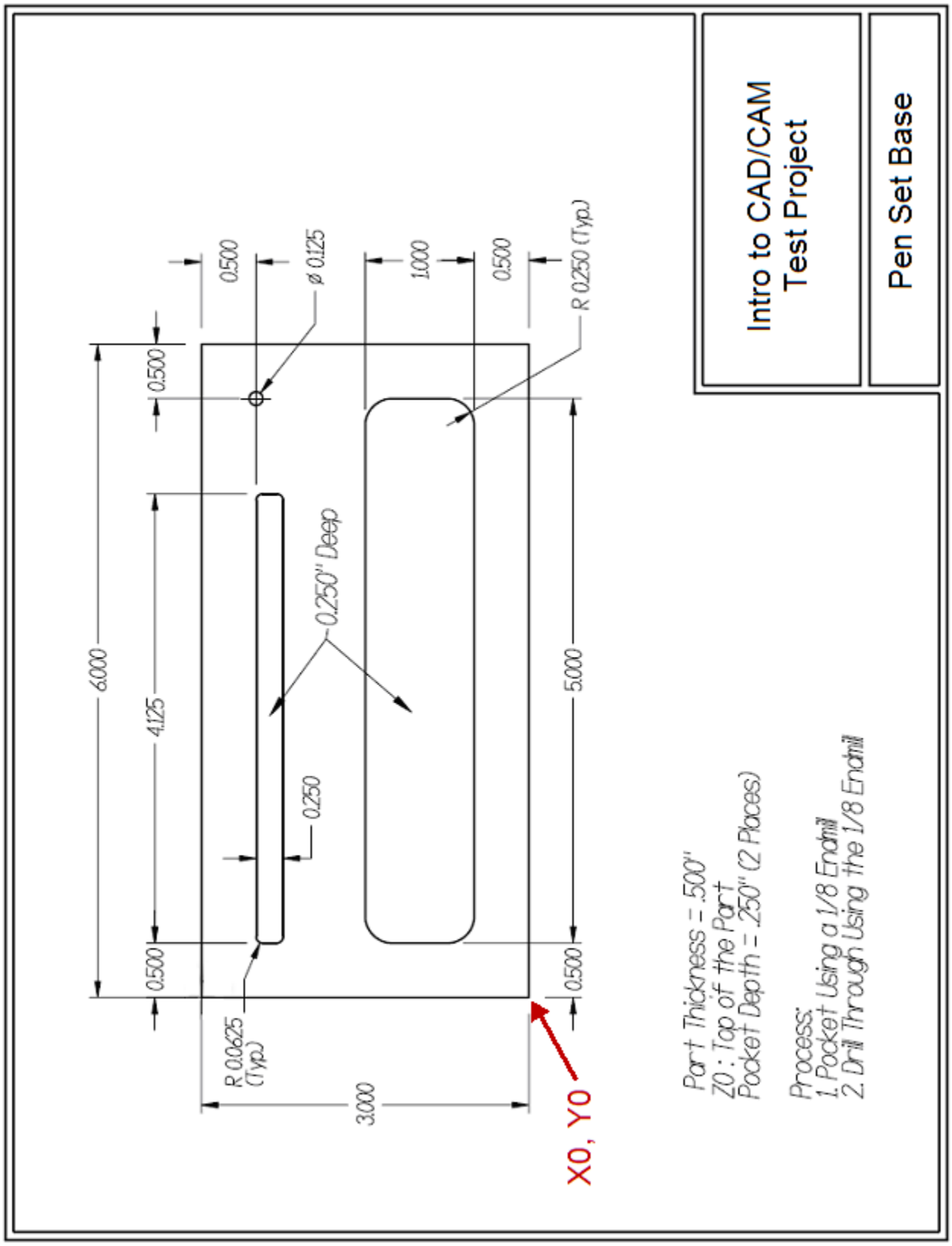
➲ **Establish Stock Setup settings:**
Stock size
Material for the part
Feed calculation

➲ **Generate a 2-dimensional milling toolpath consisting of:**
Drill toolpath
Pocket toolpath
Contour toolpath

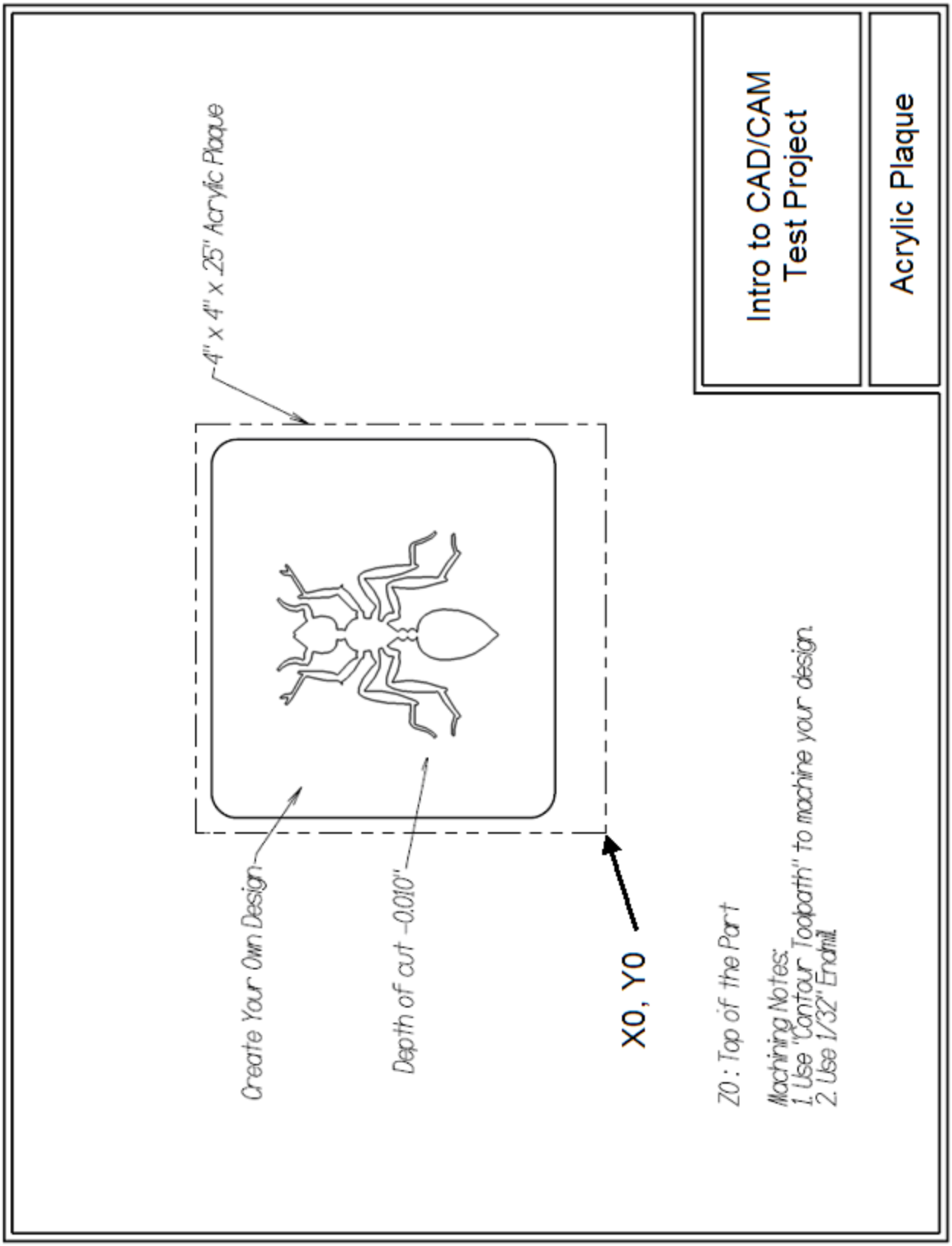
➲ **Inspect the toolpath using Mastercam's Verify and Backplot by:**
Launching the Verify function to machine the part on the screen
Using Backplot to identify the correctness of the toolpaths
Generating the NC - code

➲ **Machine the parts consisting of:**
Machine a pen base
Machine an acylic plaque

## PEN BASE DRAWING



## ACRYLIC PLAQUE DRAWING



# PEN SET PROJECT - THE PROCESS

## PEN SET BASE

### Geometry Creation

**TASK 1:** Setting the environment
**TASK 2:** Create Geometry
**TASK 3:** Save the drawing

### Toolpath Creation

**TASK 4:** Define the Stock using Stock Setup
**TASK 5:** Pocket Toolpath
**TASK 6:** Drill Toolpath
**TASK 7:** Backplot the Toolpath
**TASK 8:** Verify the Toolpath
**TASK 9:** Save the Updated MCX File
**TASK 10:** Post and Create the CNC Code File

## ACRYLIC PLAQUE

### Geometry Creation

**TASK 1:** Setting the environment
**TASK 2:** Create Geometry
**TASK 3:** Save the drawing

### Toolpath Creation

**TASK 4:** Define the Stock using Stock Setup
**TASK 5:** Contour Toolpath
**TASK 6:** Backplot the Toolpath
**TASK 7:** Verify the Toolpath
**TASK 8:** Save the Updated MCX File
**TASK 9:** Post and Create the CNC Code File

# ***PEN SET BASE***
# Geometry Creation

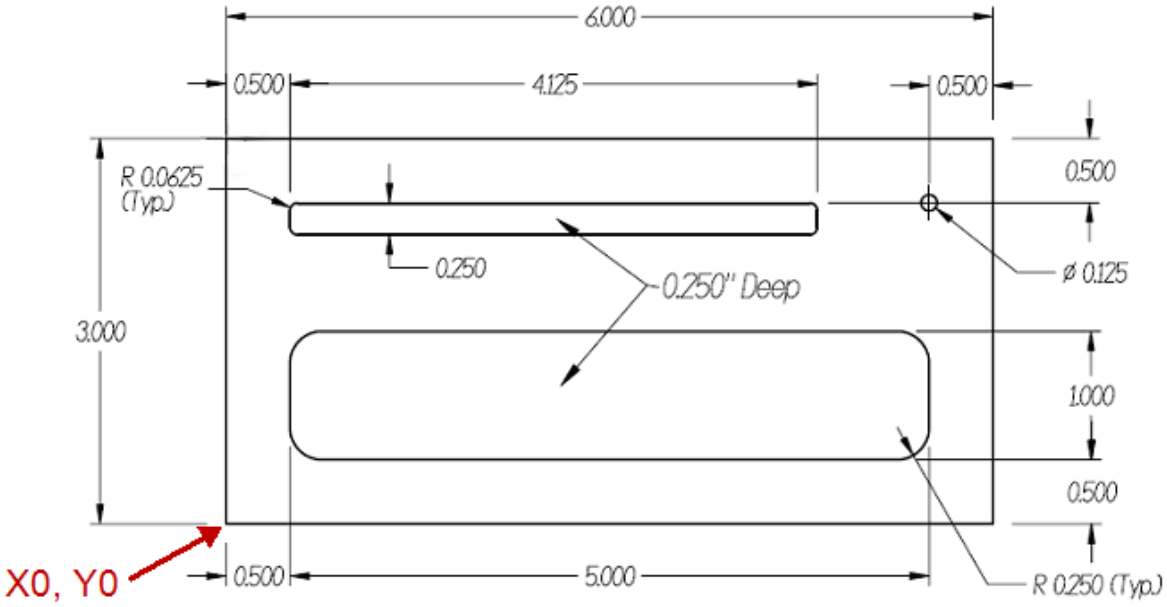
## TASK 1:
## SETTING THE ENVIRONMENT

Before starting the geometry creation you should set up the grid, toolbars and machine type Set up the Grid. This will help identify the location of the origin.

1. Customize the toolbars to machine a 2D part.
2. Set the machine.

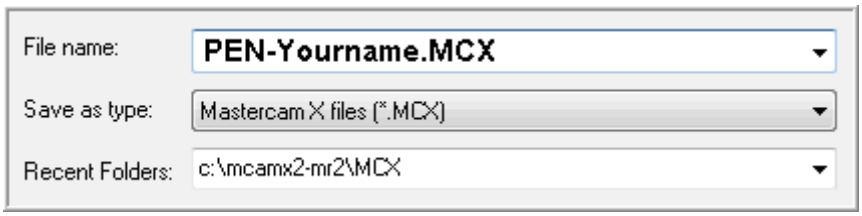
## TASK 2:
## CREATE GEOMETRY

➲ X0 Y0 is top left hand corner of the part.




## TASK 3:
## SAVE THE DRAWING

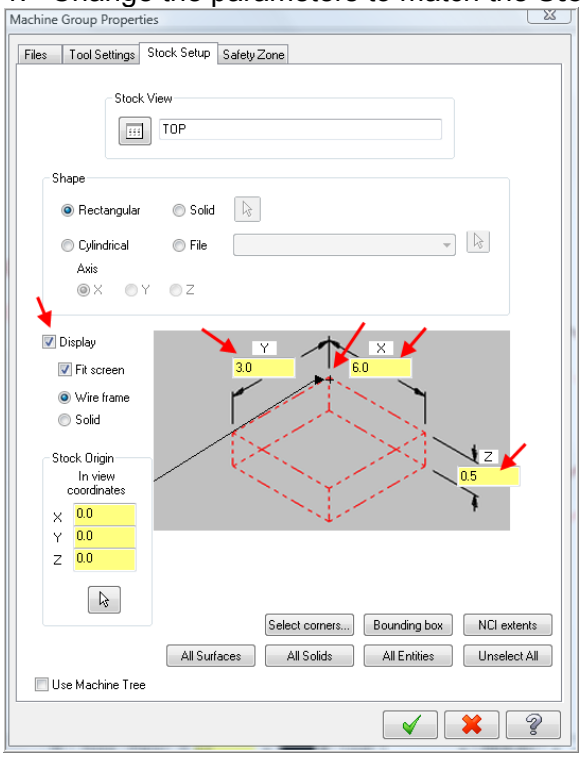
➲ **File>Save**



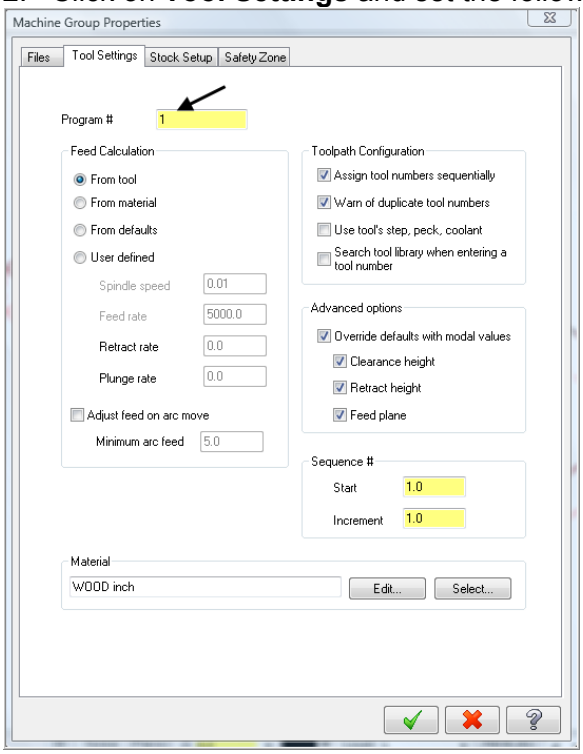
# Toolpath Creation

## TASK 4:
## DEFINING THE ROUGH STOCK USING STOCK SETUP

1. Change the parameters to match the **Stock Setup** screenshot below left:



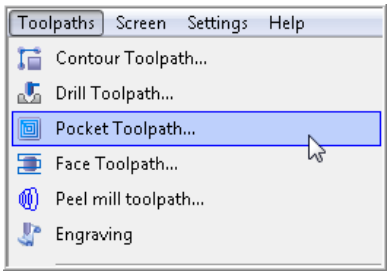

2. Click on **Tool Settings** and set the following.



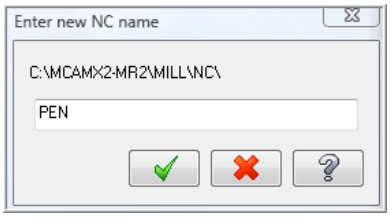
# TASK 5:
# POCKET TOOLPATH

1. Change the graphics view to a **Top View** by using the toolbar at the top of the screen.

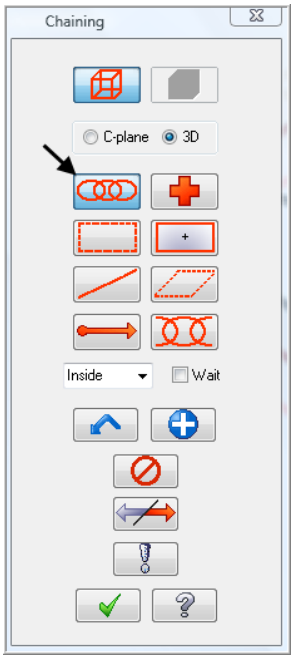
2. From the menu bar select **Toolpaths>Pocket Toolpath…**

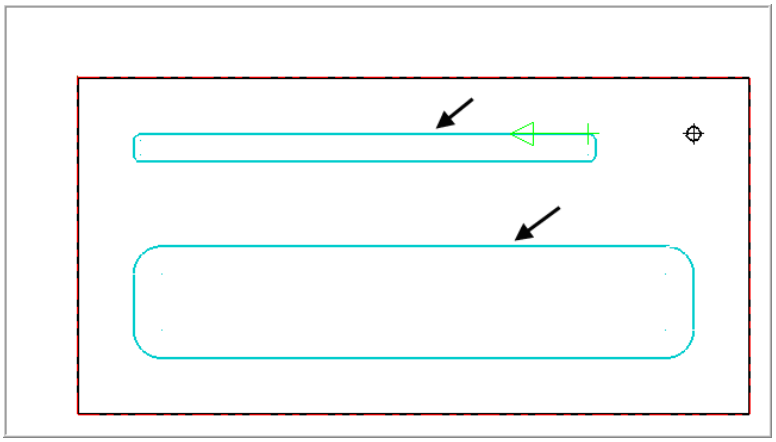



3. If prompted to **Enter new NC name** ensure **PEN** is displayed and then select the OK button  otherwise move on the step 4.

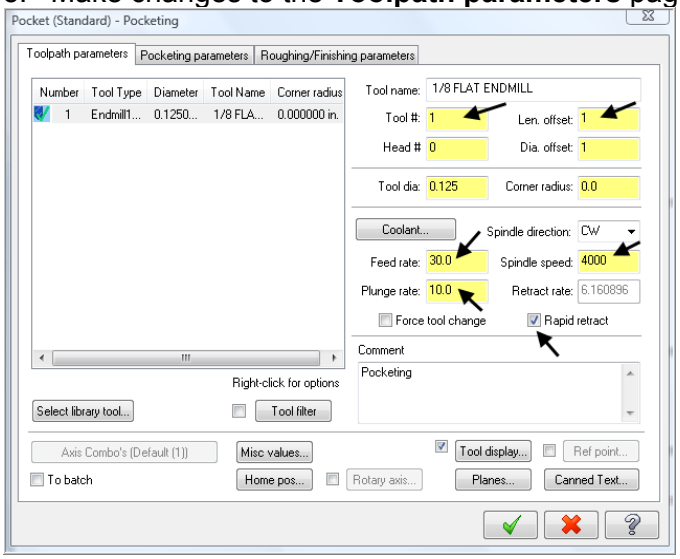



4. On the screen, you will now see the **Chaining** dialog box and a screen prompt to **Select pocket chain 1**. Select two lines as shown below right. The green arrow is pointing to the left in a counter clockwise direction, climb milling. If the arrow is not pointing to the left, reverse the direction.

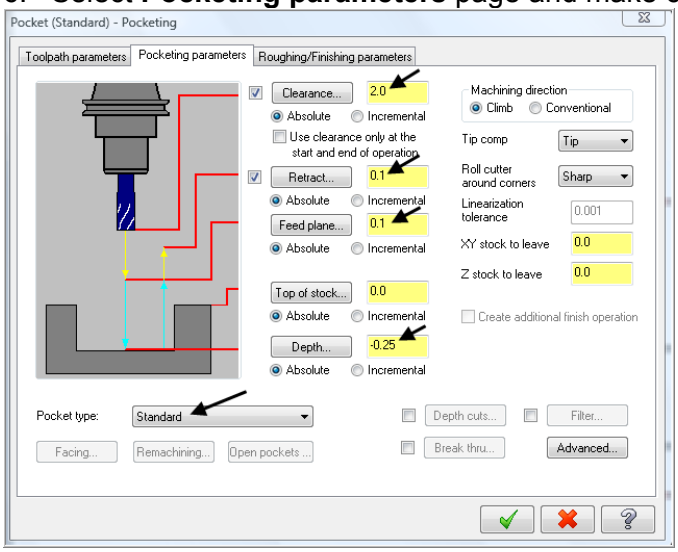


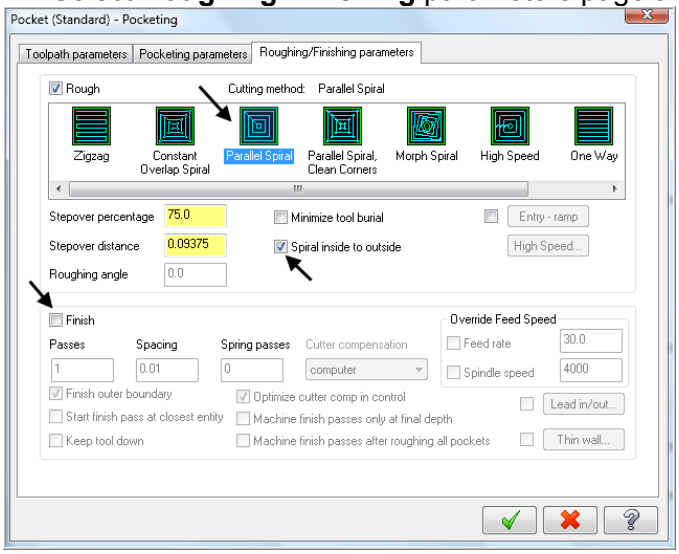
5. Make changes to the **Toolpath parameters** page as shown below.




6. Select **Pocketing parameters** page and make changes to this page as shown below.

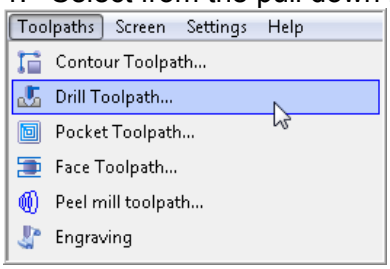



7. Select **Roughing/Finishing** parameters page and make changes as shown below.



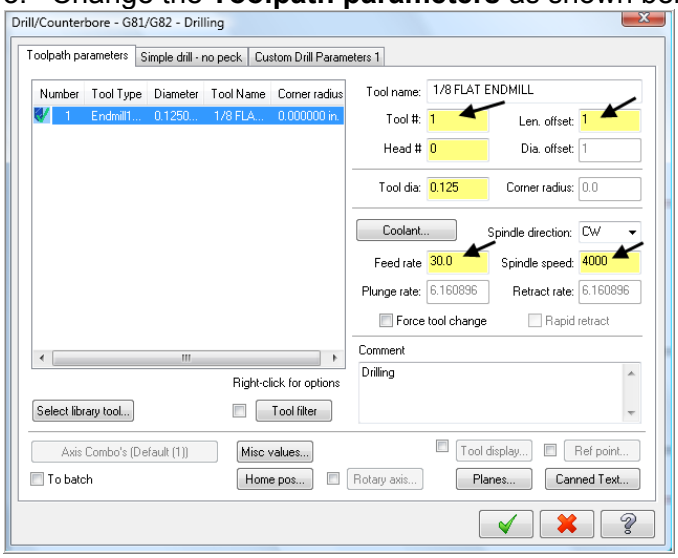
# TASK 6:
# DRILL TOOLPATH

1. Select from the pull down menu **Toolpaths>Drill Toolpath...**

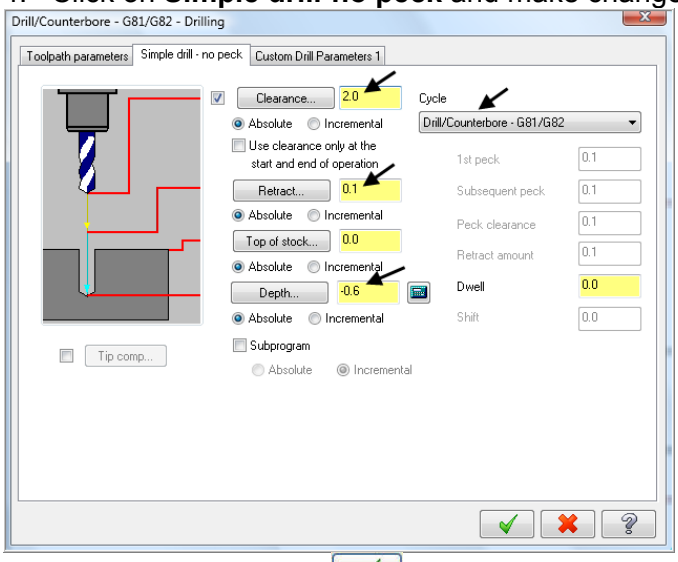



2. Select the point you wish to drill.

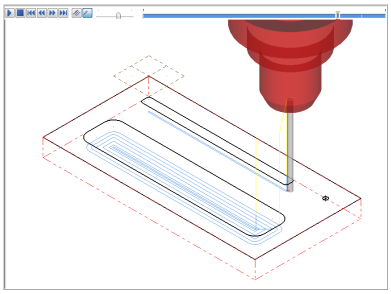
3. Change the **Toolpath parameters** as shown below.




4. Click on **Simple drill-no peck** and make changes as shown below.



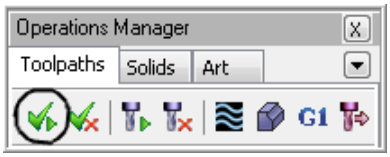

5. Select the **OK** button 



## TASK 7:
## BACKPLOT THE TOOLPATH
## TASK 8:
## VERIFY THE TOOLPATH
## TASK 9:
## SAVE THE UPDATED MCX FILE

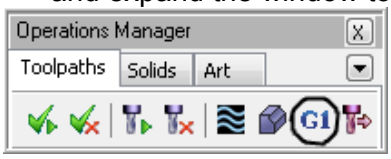
## TASK 10:
## POST AND CREATE THE CNC CODE FILE

1. Pick the **Select All** icon  from the **Toolpath manager**.

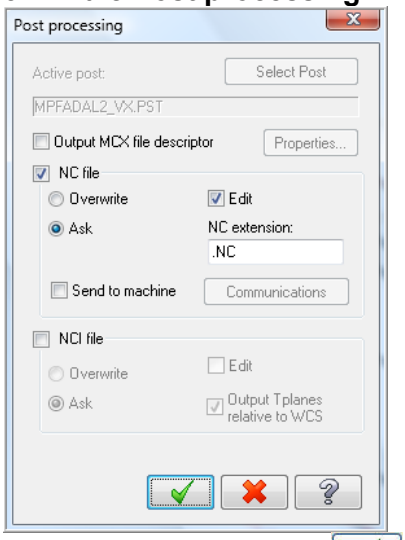



2. Select the **Post selected operations** button from the **Toolpath manager.**

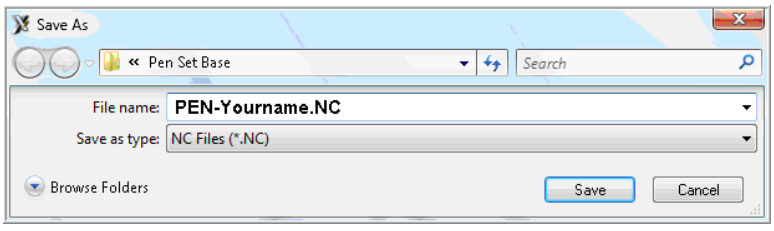
➲ **Please Note:** If you cannot see **G1** click on the right pane of the Toolpath manger window and expand the window to the right.




3. In the **Post processing** window, make the necessary changes as shown below:



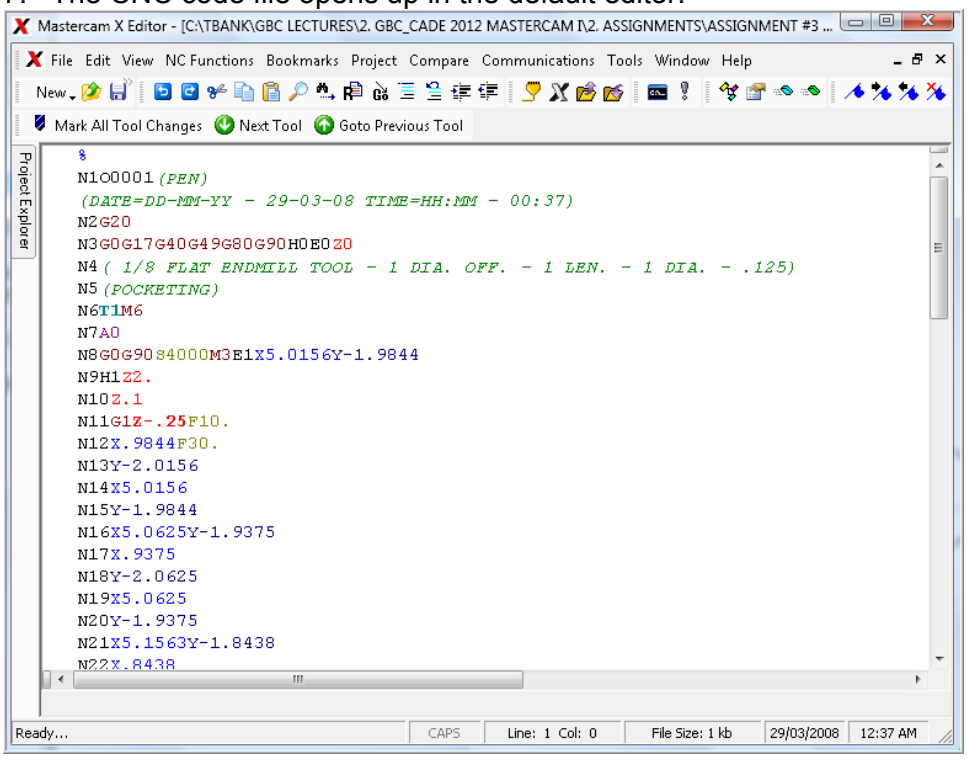

4. Select the **OK** button  to continue.
5. Ensure the same name as your Mastercam part file name is displayed in the NC File name field as shown below:



6. Select the **Save** button 

.

7. The CNC code file opens up in the default editor.




8. Select the  in the top right corner to exit the CNC editor.

9. This completes the **Pen Set Base**.

# ***ACRYLIC PLAQUE***
# Geometry Creation

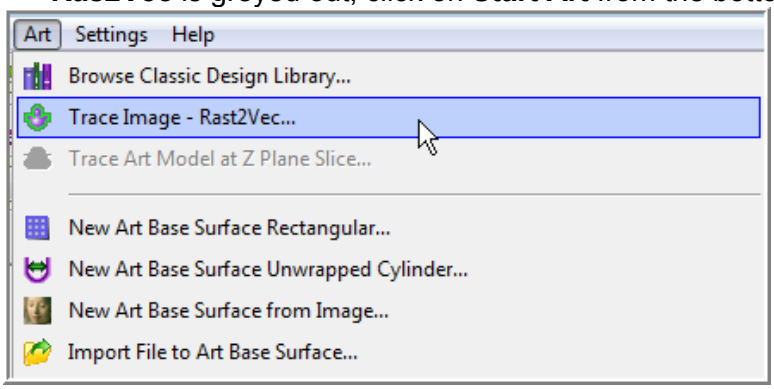
## TASK 1:
## SETTING THE ENVIRONMENT

Before starting the geometry creation you should set up the grid, toolbars and machine type Set up the Grid. This will help identify the location of the origin.

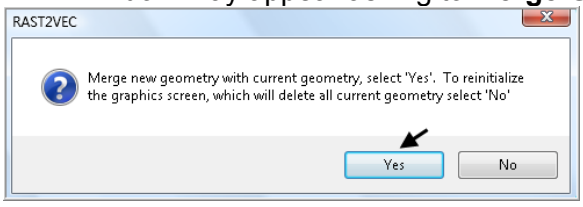
1. Customize the toolbars to machine a 2D part.
2. Set the machine type.

## TASK 2:
## CREATE GEOMETRY

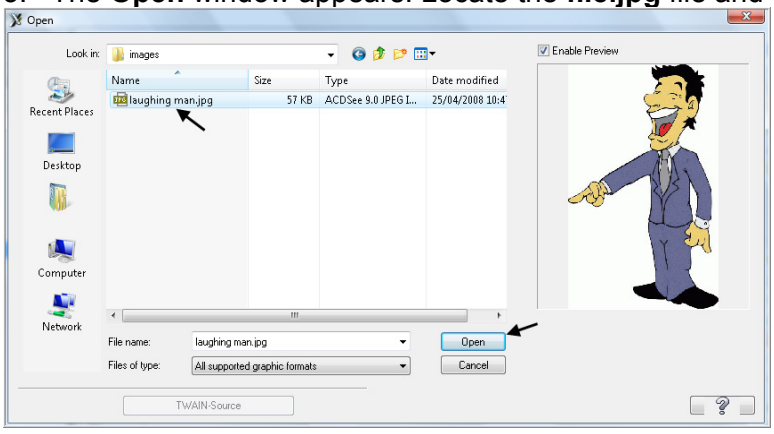
1. Select from the pull down menu **Art>Trace Image-Ras2Vec… Note:** If **Trace Image-Ras2Vec** is greyed out, click on **Start Art** from the bottom of the menu bar.




2. A window may appear asking to **Merge Geometry; Yes No**, click on Yes.




3. The **Open** window appears. Locate the **file.jpg** file and click on **Open** as shown below:

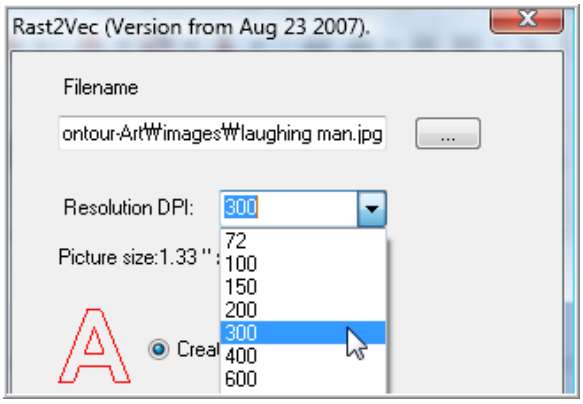


4. The **Black/White conversion** window appears. Drag the **Threshold bar** to the right until you get the image on the right pane. Notice the image gets darker. Click on **OK**  .

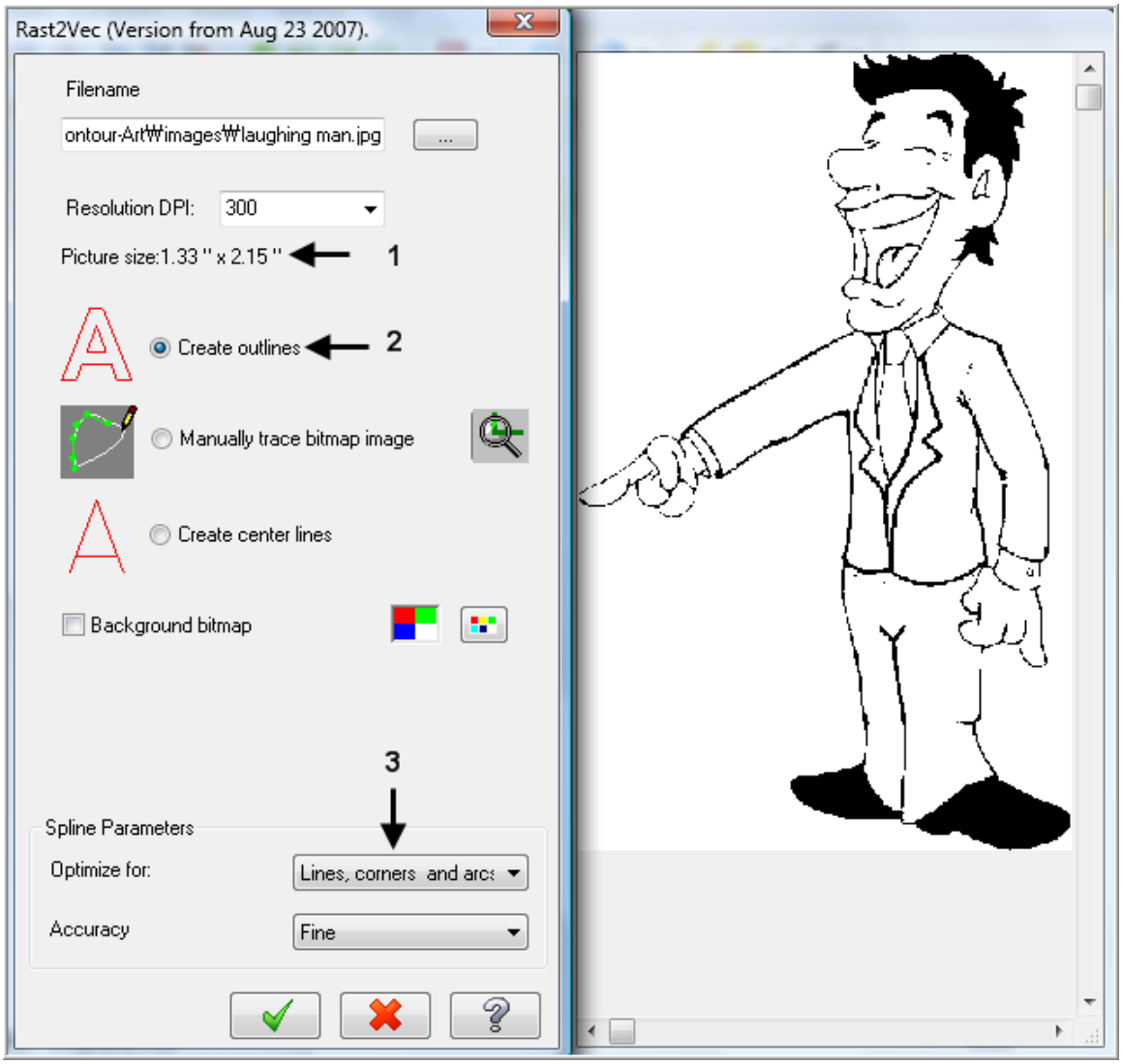

- **Linear Black/White conversion**: Converts the RGB color of each pixel to a greyscale pixel. The **Threshold** bar filters out converted pixels according to their greyscale rate.

- **Filter colors**: Filters desired colors from a graphic image for conversion. Click any color box to filter. Click the color box again to deselect. The **Tolerance** bar decides how intensely selected colors are filtered.

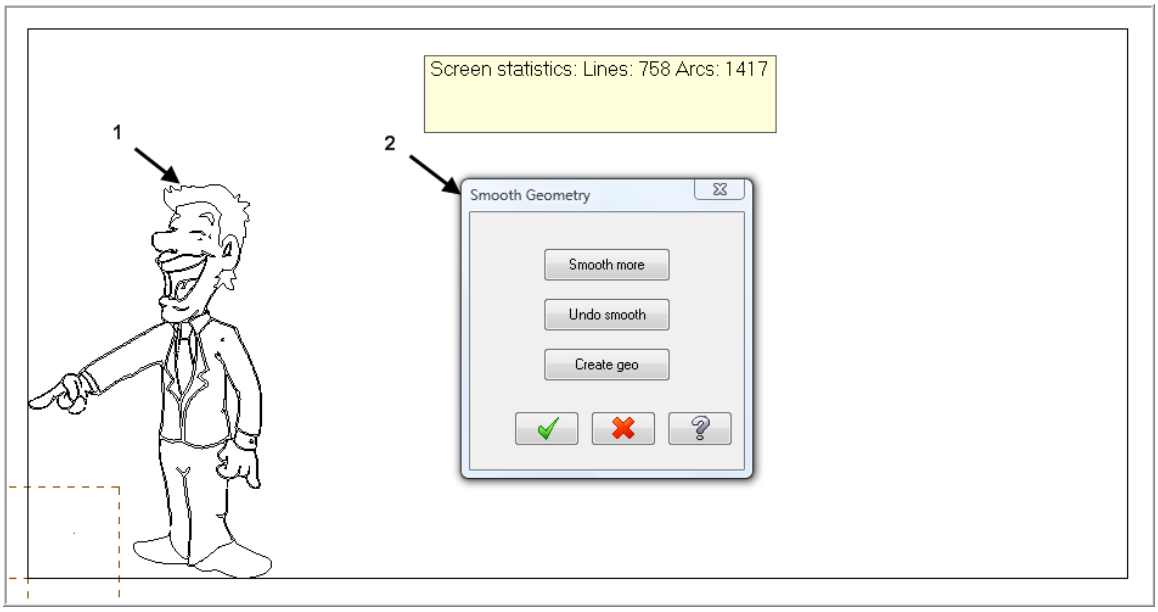
5. The **Rast2Vec** window appears. Click on the **Resolution DPI:** drop down box and select **300** as shown below:




6. Notice the picture size, **(1)** it reads **1.33" x 2.15"**. Note: when you change the **resolution** the picture size will change as well.
7. Make sure **Create Outlines (2)** is selected as shown below:
8. Select **Lines, corners and arcs** from the **Optimize for:** drop down list **(3).**
9. Click on **OK**  .

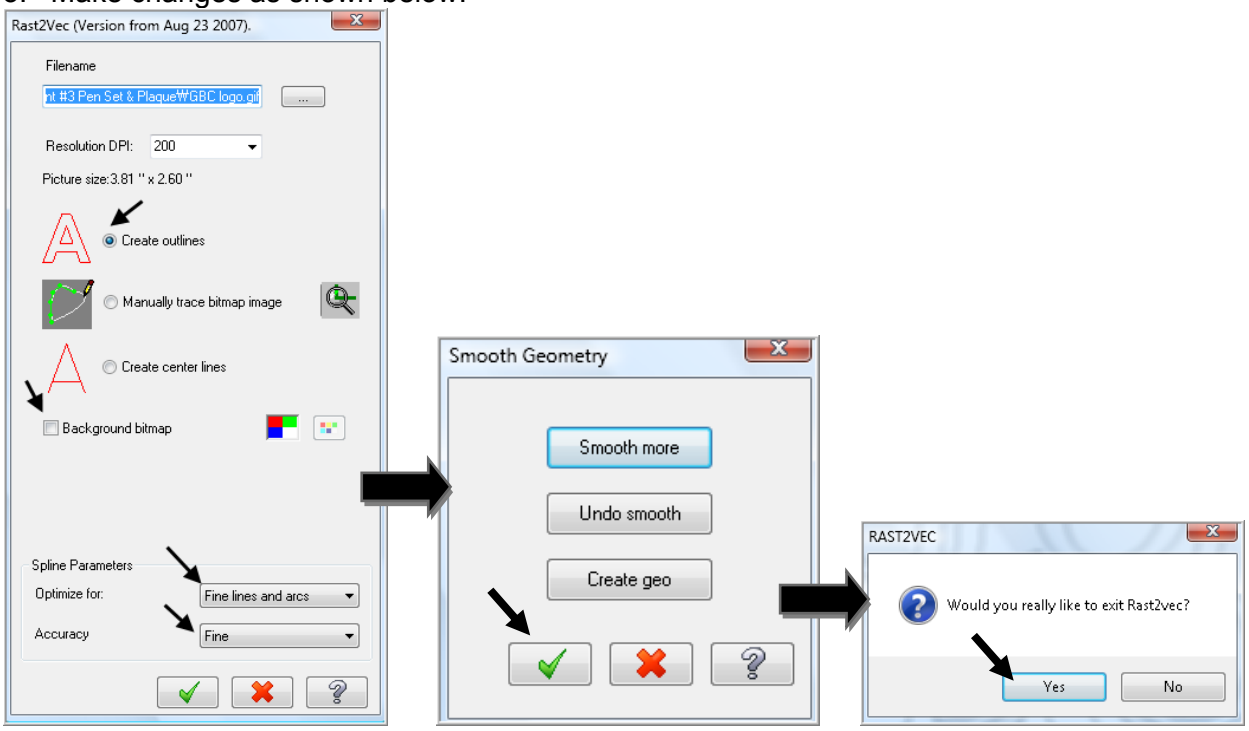


10. The **Mastercam Screen** should look like the image below: The new **image (1)** will be displayed as geometry and the **Smooth Geometry** window (2) will be appear. Click on **OK**  .




11. Click on **Yes** when asked; **"Would you really like to exit Rast2Vec?"**

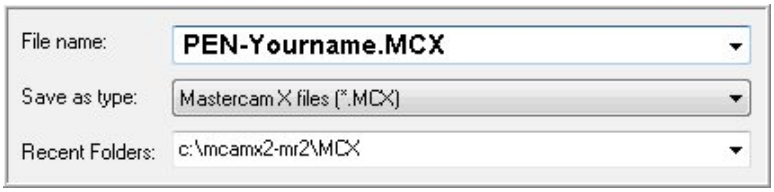
3. Make changes as shown below.




***8. Using Xform, move or modify the geometry. Ensure X0 Y0 is top left hand corner of the part.***

## TASK 3:
## SAVE THE DRAWING

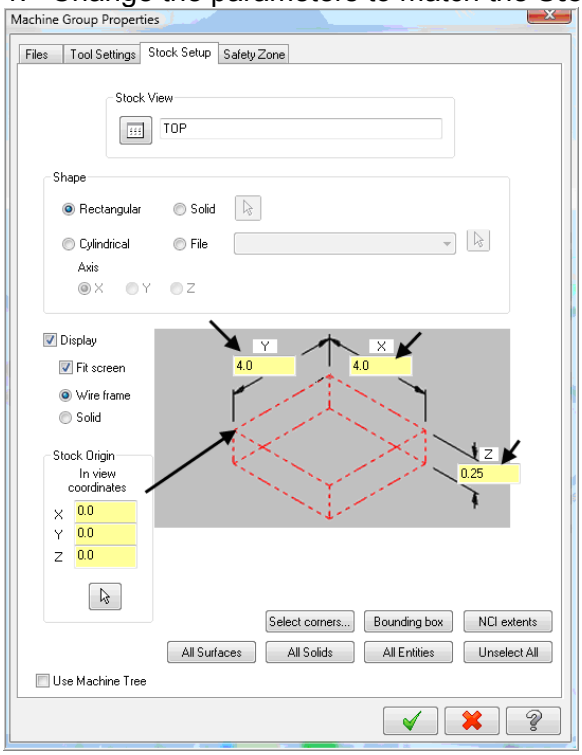
➲ **File>Save**



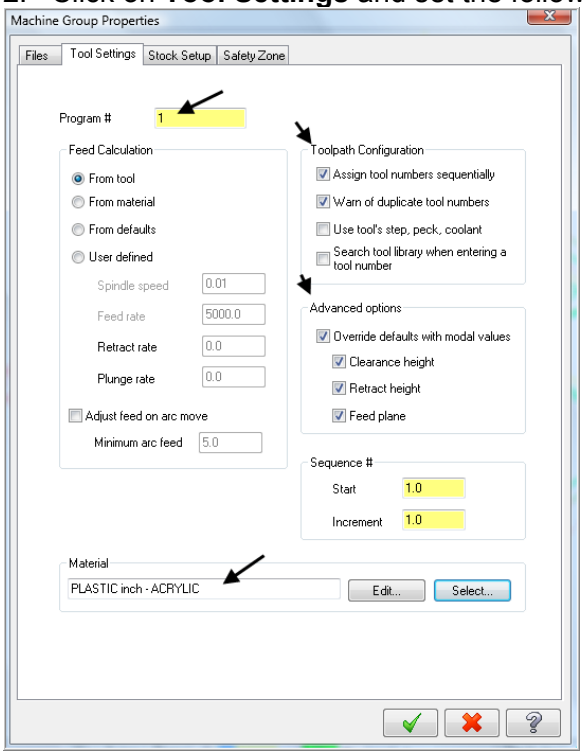

# Toolpath Creation

## TASK 4:
## DEFINING THE ROUGH STOCK USING STOCK SETUP

1. Change the parameters to match the **Stock Setup** screenshot below left:



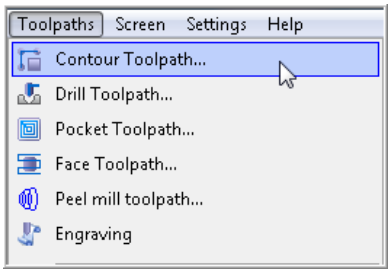
2. Click on **Tool Settings** and set the following.



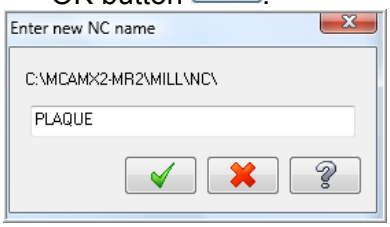
## TASK 5:
## CONTOUR TOOLPATH

1. Change the graphics view to a **Top View** by using the toolbar at the top of the screen.

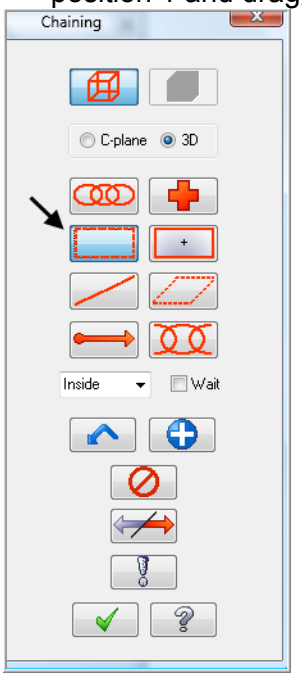
2. From the menu bar select **Toolpaths>ContourToolpath…**

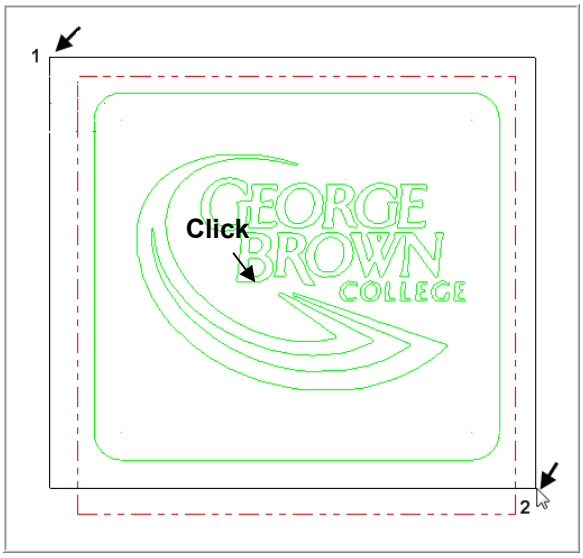



3. When prompted to **Enter new NC name** ensure **PLAQUE** is displayed and then select the OK button  .



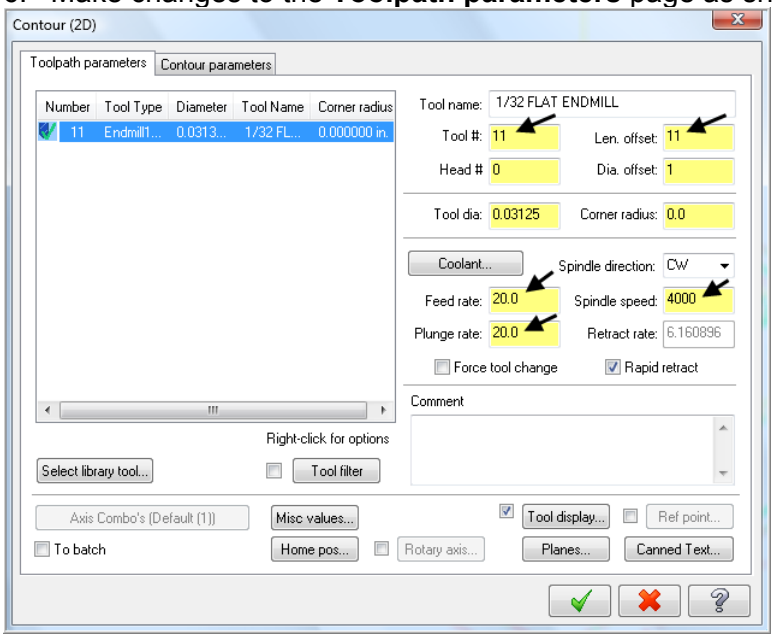

4. On the screen, you will now see the **Chaining** dialog box and a screen prompt to **Select pocket chain 1**. Create a window as shown below by selecting the upper left corner at position 1 and dragging to the opposite lower right corner at position 2.



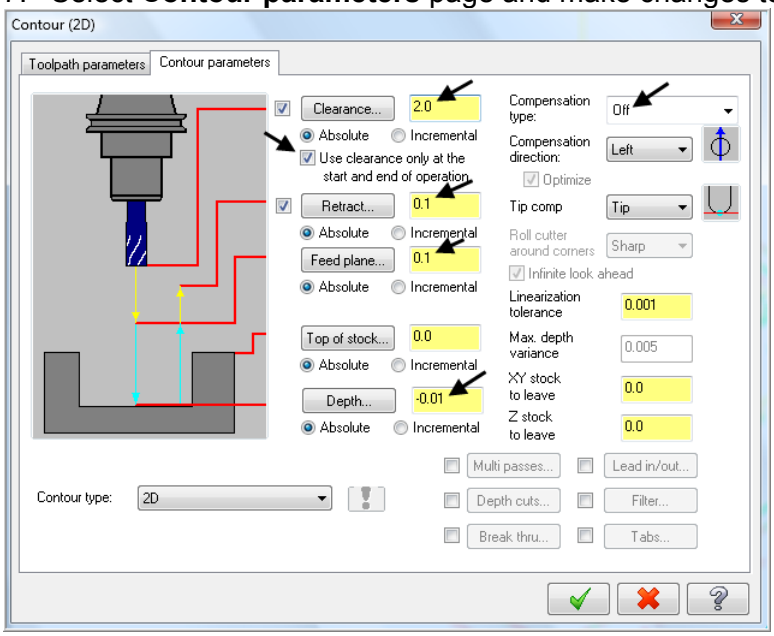




5. You will be prompted to **Sketch approximate start point**, then click inside the window box as shown above.

6. Make changes to the **Toolpath parameters** page as shown below.

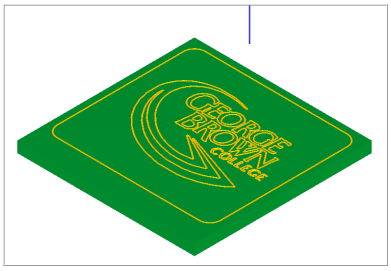



7. Select **Contour parameters** page and make changes to this page as shown below.



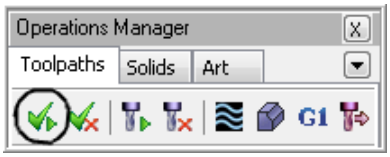

8. Select the **OK** button 

## TASK 6:
## BACKPLOT THE TOOLPATH
## TASK 7:
## VERIFY THE TOOLPATH
## TASK 8:
## SAVE THE UPDATED MCX FILE



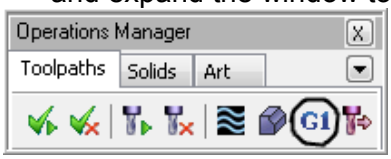
## TASK 9: POST AND CREATE THE CNC CODE FILE

1. Pick the **Select All** icon from the **Toolpath manager**.

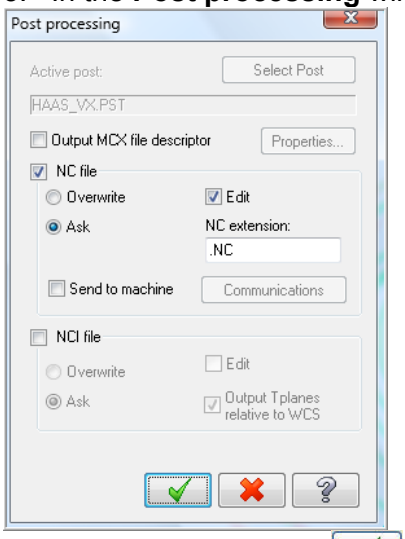



2. Select the **Post selected operations** button from the **Toolpath manager.**

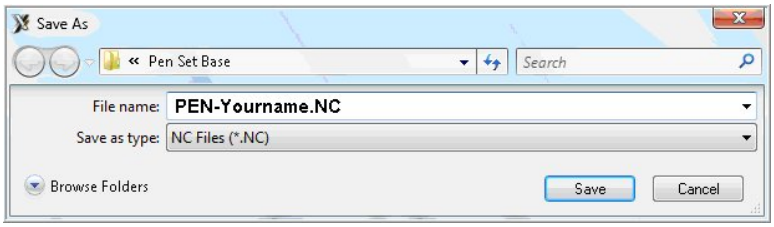
➲ **Please Note:** If you cannot see **G1** click on the right pane of the Toolpath manger window and expand the window to the right.




3. In the **Post processing** window, make the necessary changes as shown below:

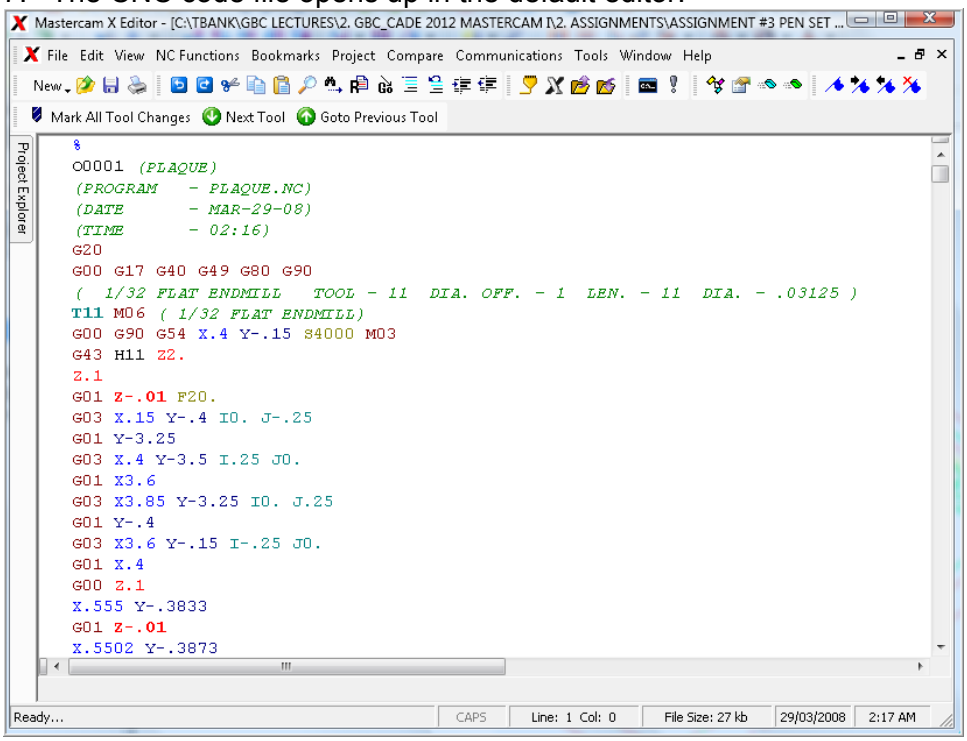



4. Select the **OK** button to continue.
5. Ensure the same name as your Mastercam part file name is displayed in the NC File name field as shown below:



6. Select the **Save** button 

.
7. The CNC code file opens up in the default editor.




8. Select the  in the top right corner to exit the CNC editor.
9. This completes **Acrylic Plaque**.

# Pen Set Base
# Check List

| No | Items to be checked | | Check |
|---|---|---|---|
| | **General** | | |
| 1 | **Machine Type:** | Check with your instructor | |
| 2 | **Program Number:** | 1 | |
| 3 | **Program Name:** | PEN-Yourname.MCX | |
| 4 | **Location of X0 Y0 :** | Bottom left hand corner of material | |
| 5 | **Z=0:** | Top of the part | |
| 6 | **Stock Setup:** | Material size 6” x 3” x .5” | |
| 7 | **Cutter:** | Ø.125 Endmill- Tool # 1 & Length offset #1 | |
| | **Drilling** | | |
| 8 | **Feedrate:** | 30 inches per minute | |
| | **Spindle speed:** | 4000 rpm / Coolant Off | |
| 9 | **Depth of Hole:** | -0.6” in Absolute | |
| 10 | **Clearance:**<br>**Retract:**<br>**Top of Stock:** | 2 in.<br>0.1 in.<br>0 (all in absolute) | |
| | **Pocketing** | | |
| 11 | **Feedrate:** | 30 inches per minute | |
| 12 | **Plunge feedrate:** | 10 inches per minute | |
| 13 | **Spindle speed:** | 4000 rpm | |
| 14 | **Pocket Depth:** | -.25” in Absolute | |
| 15 | **Clearance:** | 2 in. | |
| 16 | **Retract & Feed plane:** | 0.1 in. Top of Stock 0 (all in absolute) | |
| 17 | **Pocket Cutting Method:** | Parallel Spiral ( NO FINISH) | |
| 18 | **Check mark on:** | “Spiral Inside to Outside” | |
| | **Posting** | | |
| 20 | **Save the CNC code file:** | on a diskette or an USB | |
| 21 | **File Name:** | PEN-Yourname.NC | |
| | **Machine Part** | | |
| 22 | Prepare material for machining | | |
| 23 | Fixture material onto Machine table | | |
| 24 | Install proper cutter into Machine Spindle | | |
| 25 | Set up Work and Tool Offsets on Machine | | |
| 26 | Load proper NC file | | |
| 27 | Machine part | | |
| 28 | Deburr | | |
| 29 | Clean machine and put tools back | | |

# 4 x 4 Acrylic Plaque
# Check List

| No | Items to be checked | | Check |
|---|---|---|---|
| | **General** | | |
| 1 | **Machine Type:** | Check with Instructor | |
| 2 | **Program Number:** | #1 | |
| 3 | **Location of X0 Y0:** | Bottom left hand corner of material (both geometry & stock) | |
| 4 | **Z=0:** | is the top of the part | |
| 5 | **Stock Setup:** | Material size 4” x 4” x .25” (Material type doesn’t matter) | |
| 6 | **Cutter:** | Ø1/32 (Ø.03125 Endmill- Tool # 2 & Length offset #2) | |
| | **Contouring** | | |
| 7 | **Feed Rate:** | 20 inches per minute | |
| 8 | **Plunge Rate:** | 20 inches per minute | |
| 9 | **Retract Rate:** | Rapid – Check mark for Rapid Retract | |
| 10 | **Spindle Speed:** | 4000 rpm | |
| 11 | **Depth of cut:** | -0.010” in Absolute | |
| 12 | **Clearance:** | 2.0 Check mark for “Clearance only at start and end of operation” | |
| 13 | **Retract:** | 0.1 in Absolute | |
| 14 | **Feed Plane:** | 0.1 in Absolute | |
| 15 | **Top of Stock:** | 0 in Absolute | |
| 17 | **Compensation Type:** | OFF | |
| | **Posting** | | |
| 19 | **Save the CNC code file:** | on a diskette or an USB | |
| 20 | **File Name:** | PLAQUE-Yourname.NC | |
| | **Machine Part** | | |
| 1 | Prepare material for machining | | |
| 2 | Fixture material onto Machine table | | |
| 3 | Install proper cutter into Machine Spindle | | |
| 4 | Set up Work and Tool Offsets on Machine | | |
| 5 | Load proper NC file | | |
| 6 | Machine part | | |
| 7 | Deburr | | |
| 8 | Clean machine and put tools back | | |