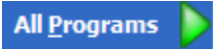# STATE QUARTER DISPLAY

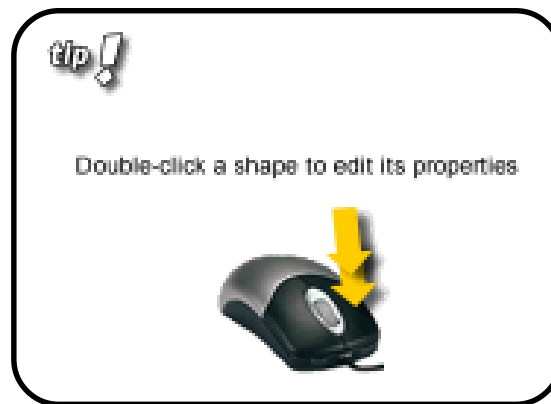## Section 1 - Starting the QuickCAM 2D Software

1. OPEN **QuickCAM 2D**.
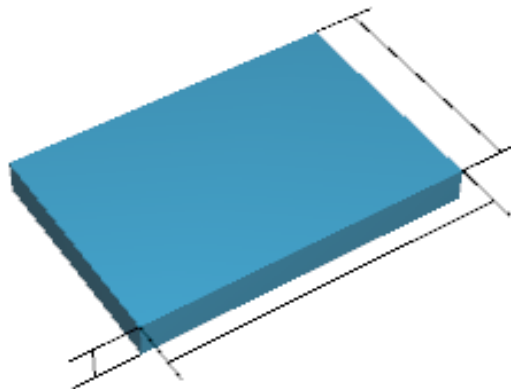
   - Click on the **START** start menu.

   - Select **All Program**s All Programs .

   - Select **Denford**.

   - Finally select **QuickCAM 2D Designer**.

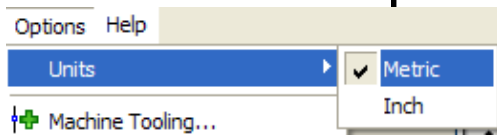2. A **TIPS menu** may appear, if it does *read the tip* then *click* **OK** to continue.

"Billet" *is the term for the block of material being machined or cut!*

3. The **Billet Dimensions menu** will appear - make the following changes and *click* **OK**.

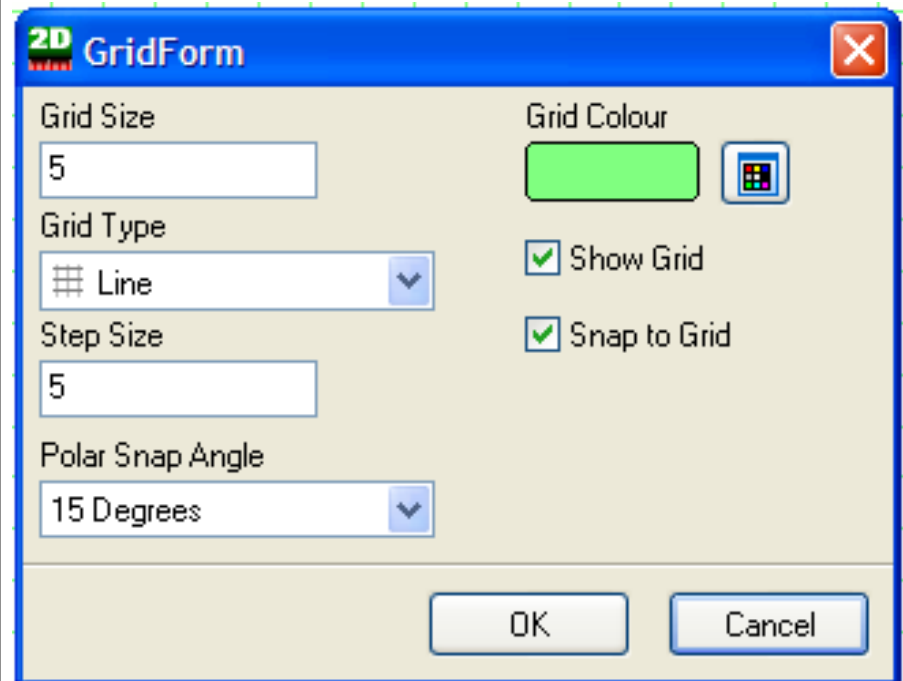## Section 2 - Setting up the Drawing Parameters

1. Go to the **OPTIONS menu** at the top of the screen.

    • Select **Units**.

    • Select **Metric**.

2. Again, go to the **OPTIONS menu** at the top of the screen.

    • Select **GRID** Grid... .

3. When the **Grid Form** appears, make the following changes ..

2D GridForm

Grid Size

5

Grid Type

⊞ Line

Step Size

5

Polar Snap Angle

15 Degrees

Grid Colour

Show Grid ✓

Snap to Grid ✓

OK   Cancel

    • When finished making the changes, *click* **OK.**

4. You are now ready to start designing!

GOOD LUCK!

Section 3 - Creating Your Design

1. *Select* the **RECTANGLE tool** from the toolbar on the left side of the screen.

2. Move the cursor until the coordinates at the bottom of the screen read ... `X: 5.000` `Y: 5.000` .

3. *Click and Drag* the cursor to the upper right until you have a rectangle drawn **180mm X 120mm.**

   `X: 185.000` `Y: 125.000` `Start X: 5.00 Y: 5.00 Width 180.00 Height 120.00`

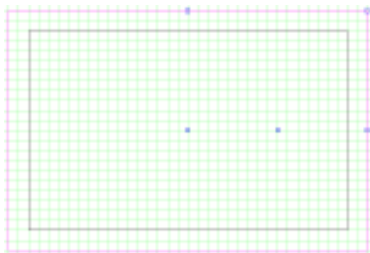   • Click the mouse button and a rectangle will appear.

4. While the RECTANGLE tool is still selected, move the cursor to the coordinates ... `X: 15.000` `Y: 15.000` .

5. *Click and Drag* the cursor to the upper right until you have a rectangle drawn **160mm X 100mm.**

   `X: 175.000` `Y: 115.000` `Start X: 15.00 Y: 15.00 Width 160.00 Height 100.00`

6. Choose the **SELECT tool** from the toolbar on the left.

   • *Click* on one of the rectangles you just created.

   • After *clicking* on the rectangle you will notice a change in it's appearance. *See to the left.*

   • This indicates the object is now *selected* and that it can be **edited**.

7. *Right-Click* on the rectangle that is selected.

   • When the **EDIT menu** appears, *select* **Properties**.

   • Change the **Corner Radius** to 10.000mm and *click* **OK**.
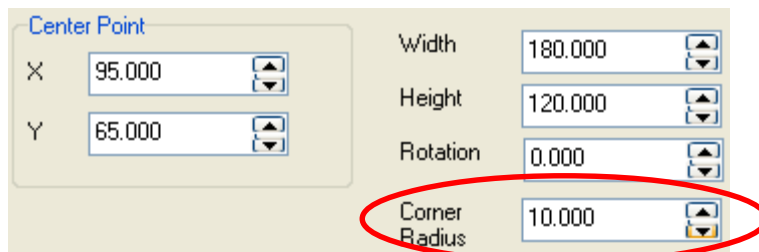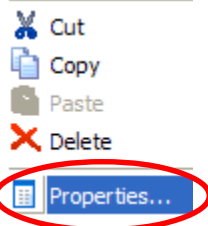
8. Your design should now resemble the image below.
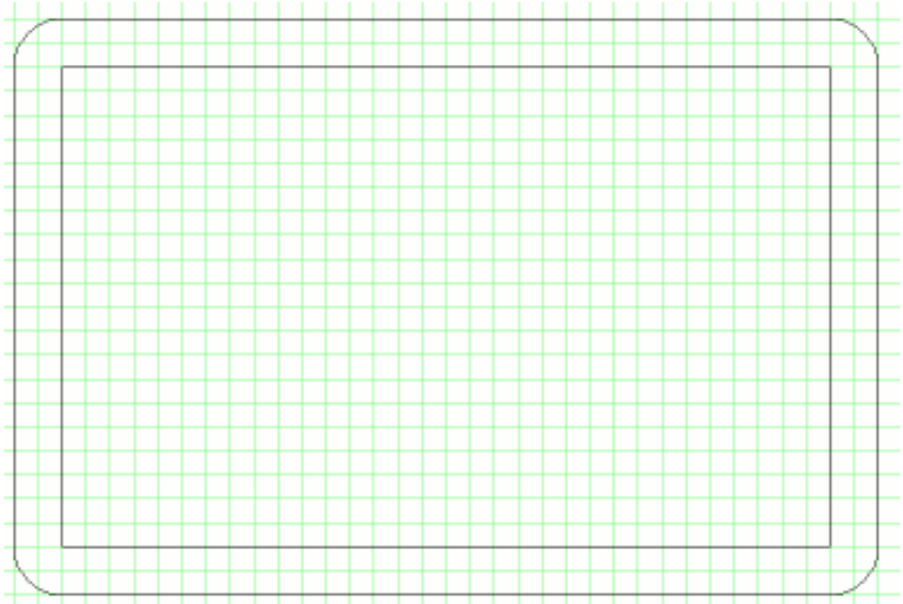
9. **Repeat** the above processes and radius the corners on the **second rectangle**!
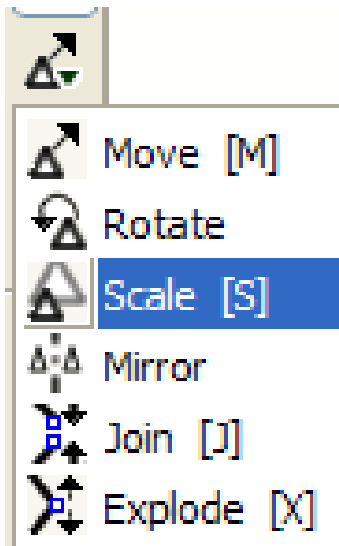
10. From the top menu select **FILE**, then **IMPORT**.

- Make sure the directory is set to ...
  My Documents/Denford/Drawing Files.

- Choose **MT** and *click* **OPEN**.
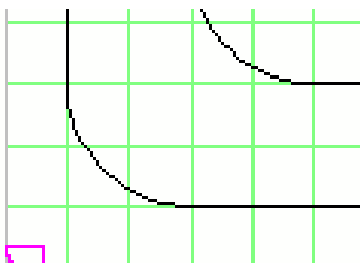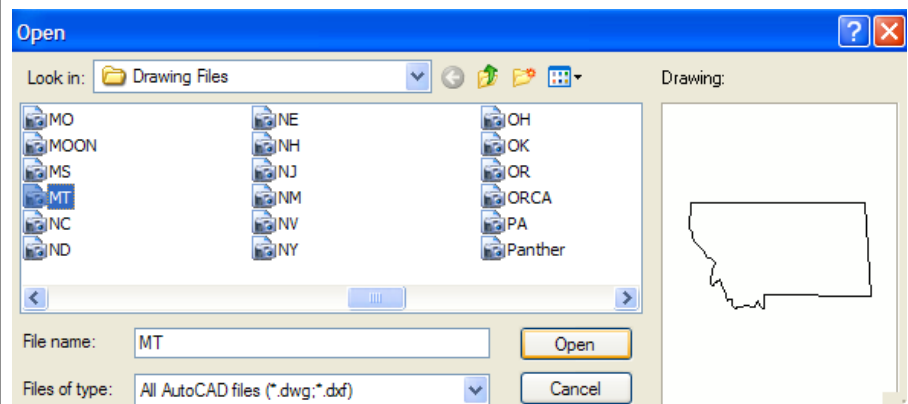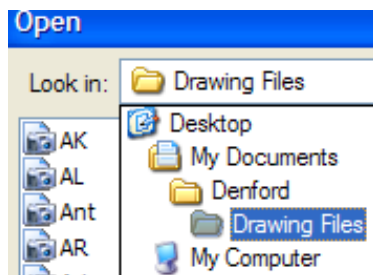
- **Where is the image at?**

- **Don't worry it's located at the bottom left corner of the drawing area.** *See to the left!*

Click and hold to make the DROP-DOWN menu appear!

11. *Select* the **SCALE tool** from the toolbar on the left side of the screen OR **press S on the keyboard**.

- You will be asked to **SPECIFY BASE POINT** ...

- Click in the lower left corner of the drawing area.

- Move the cursor to the **Scale Factor box** located at the top of the screen.

- Enter **45%** and *click* **OK**. *See example below!*

12. The outline of MONTANA if now the **correct scale**, but it has to be **moved** to fit in the center of the rectangles.

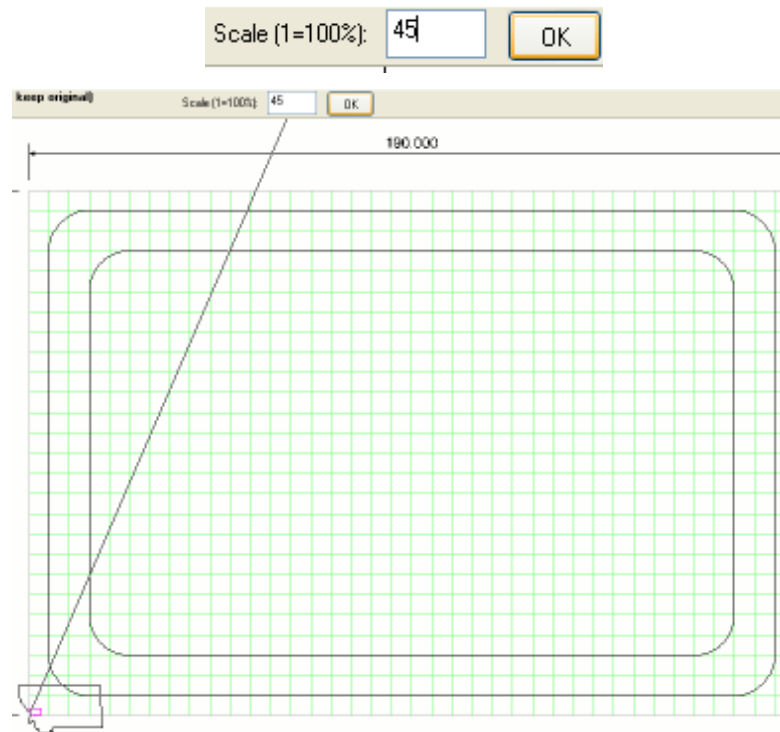- Use the **SELECT tool** 🔀 to highlight the outline of Montana.

- Select the **MOVE tool** from the toolbar on the   left side of the screen OR **press M on the keyboard**.

- You will be asked to **Specify Base Point or Displacement** ...

MOVE: Specify base point or displacement

- *Click* on the upper left corner of the Montana Outline.

- *Drag* the image until it 's approximately centered in the rectangles.

- When in the correct location ... **CLICK**. *See example below!*

13. Before going any further, you MUST use the **SAVE** command located in the FILE menu at the top of the screen

  - Make sure you are saving in the QUICKCAM2D Directory!

  - Save the file using the following format ...
    <Your Initials> State Qrt (example: CBT State Qrt).

Click and hold to make the
DROP-DOWN menu
appear!

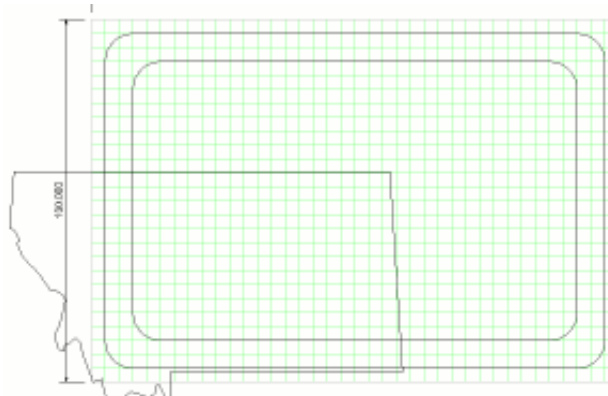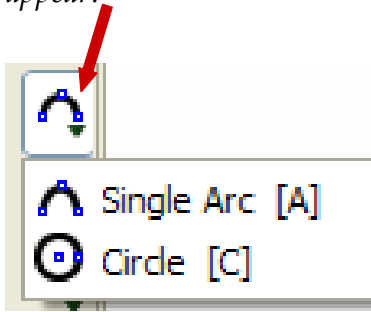14. Select the CIRCLE tool from the tool on the left side of the screen OR **press C on the keyboard**.

- If the **ADD ARC tool** appears in the toolbar instead of the **Circle tool**, *click on and hold* until the drop down menu appears.

15. Move the cursor on to the screen and **draw 2 circles** as *shown below!*

- At this time **DON'T worry** about size or location.
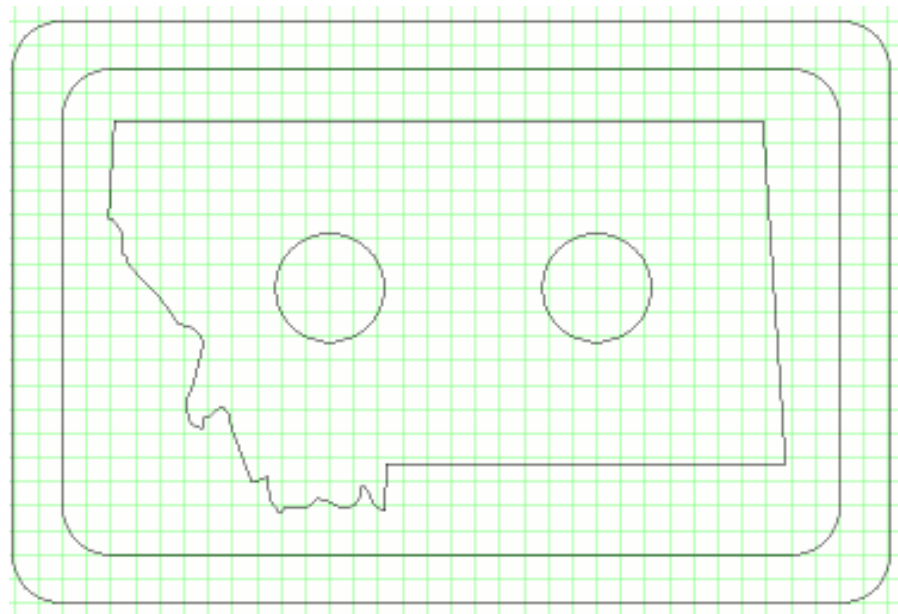
16. Use the **SELECT tool** to highlight the <u>left circle</u>.

- R*ight-click* on the left circle and *select* **PROPERTIES**.

- Make the following changes, *click* **APPLY**, then **OK**.

17. Use the **SELECT tool** to highlight the <u>right circle</u>.

- R*ight-click* on the right circle and *select* **PROPERTIES**.

- Make the following changes, *click* **APPLY**, then **OK**.

Edit Circle

Radius 12.050
Diameter 24.100

Center Point
X 125.000
Y 75.000

OK   Cancel   Apply
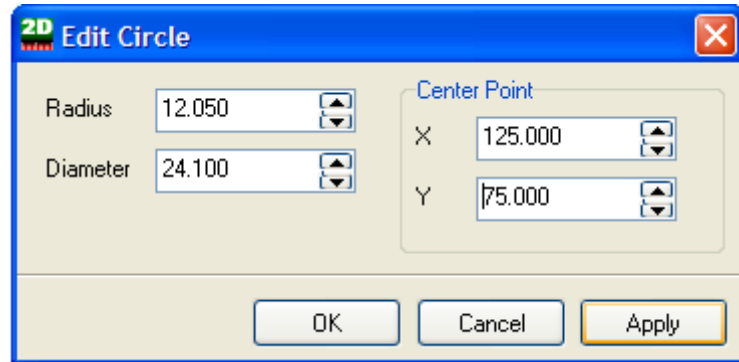
SAVE YOUR WORK!
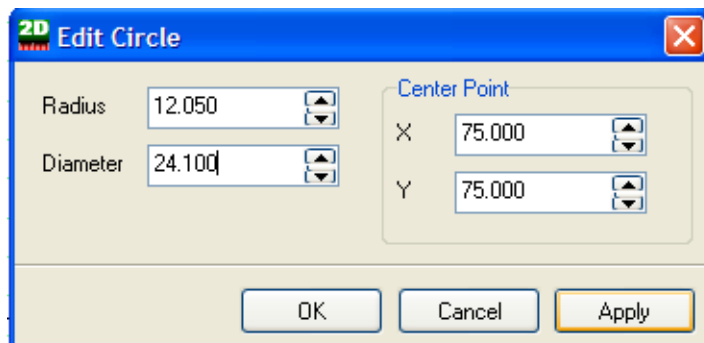
18. Select the TEXT tool **A** from the tool on the left side of the screen OR **press T on the keyboard**.

- Move the cursor to coordinates **X: 55.000, Y: 40.000**.

- *Click and drag* the cursor to **X: 145.000, Y: 60.000**.

- When the window below appears make the following changes, then *click* **OK**.

Text   Montana

Font   isocp
Alignment   Left   A a B b Y y Z x
Height   20.000
Rotation   0.000   Insertion Point
Width Factor   1.000   X: 55.000
Oblique Angle   0.000   Y: 40.000

☐ Upside Down   ☐ Backwards

OK   Cancel   Apply
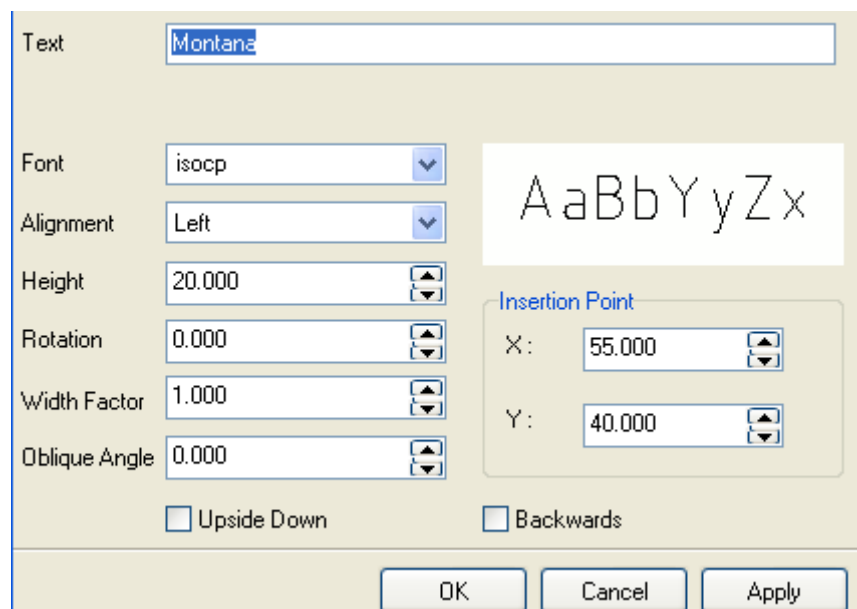
19. Select the TEXT tool **A** from the tool on the left side of the screen OR **press T on the keyboard**.

- Move the cursor to coordinates **X: 65.000, Y: 90.000**.

- *Click and drag* the cursor to **X: 135.000, Y: 100.000**.

- When the window below appears make the following changes, then *click* **OK**.
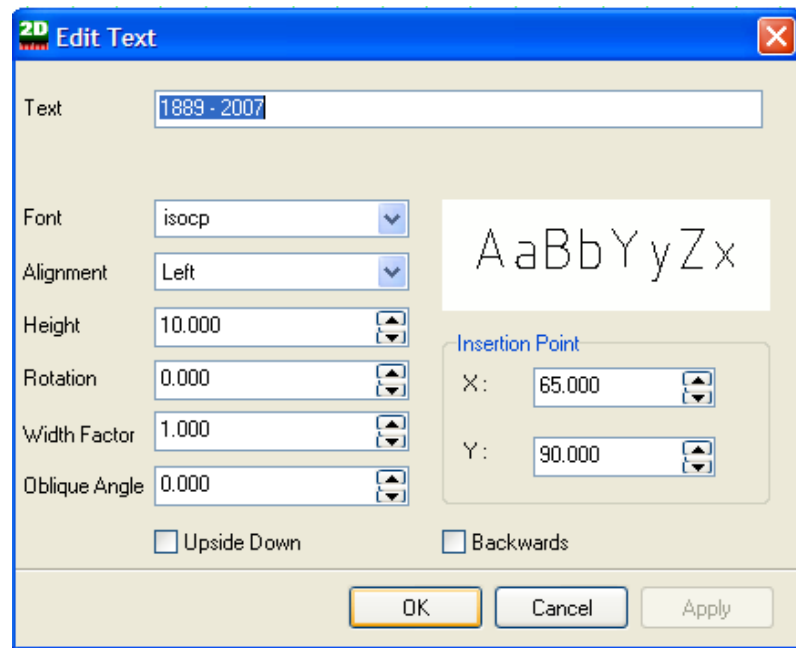
2D Edit Text

Text	1889 - 2007

Font	isocp		AaBbYyZx
Alignment	Left		
Height	10.000		Insertion Point
Rotation	0.000		X: 65.000
Width Factor	1.000		Y: 90.000
Oblique Angle	0.000		

☐ Upside Down   ☐ Backwards

[OK]   [Cancel]   [Apply]

SAVE YOUR WORK!

20. Your **completed design** should look like the image below!
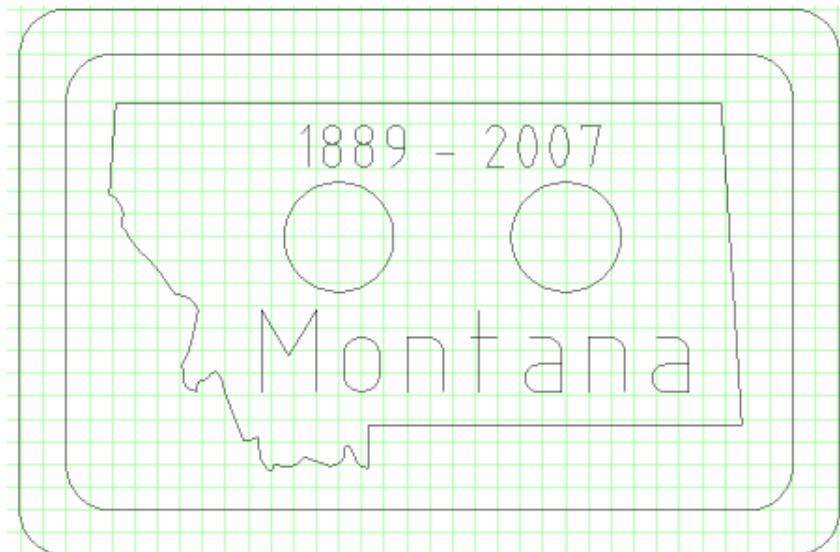
Section 4 - Using the CAM Wizard

1. *Select* the **CAM Wizard** located at the top right of the screen.
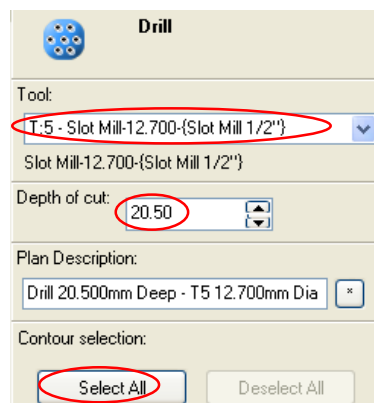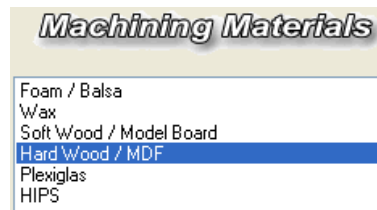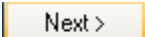
   - When the Machining Materials window appears, *select* **Hard Wood/MDF**.

   - Click **NEXT** Next > to continue on!

2. To add a **DRILL Machining Plan**, *click* on Drill.

   Drill

3. Make the following selections ...

   - Tool: T:5 - **Slot Mill-12.700-{Slot Mill 1/2"}**
   - Depth of cut: **20.50**
   - Contour Selection: **Select ALL**

4. *Click* on **APPLY,** then *click* **OK**.

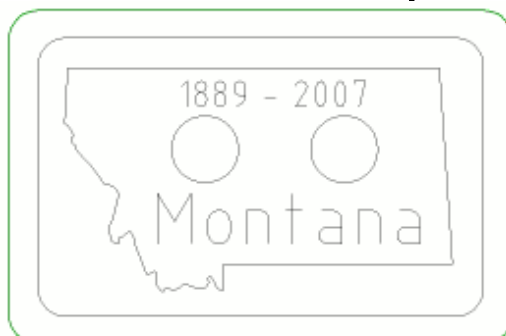5. Add a **Outside Offset Machining Plan**, *click* on Outside Offset.

   Outside Offset

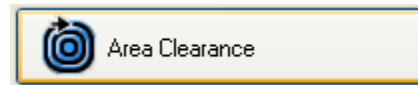6. Make the following selections ...

   - Tool: T:5 - **Slot Mill-12.700-{Slot Mill 1/2"}**
   - Depth of cut: **20.50**

7. Move the cursor over the line shown below left... then click on it!

   - If the line color didn't change to green, see your teacher now!

   - *Click* on **APPLY,** the *click* **OK**.

8. Add a **Area Clearance Machining Plan**, *click* on Area Clearance.

Make the following selections ...

- Tool: T:5 - **Slot Mill-12.700-{Slot Mill 1/2"}**
- Depth of cut: **3.00**
- Stepover: **50%**

9. Move the cursor over the line shown below... then click on it!

- If the line color didn't change to green, see your teacher now!
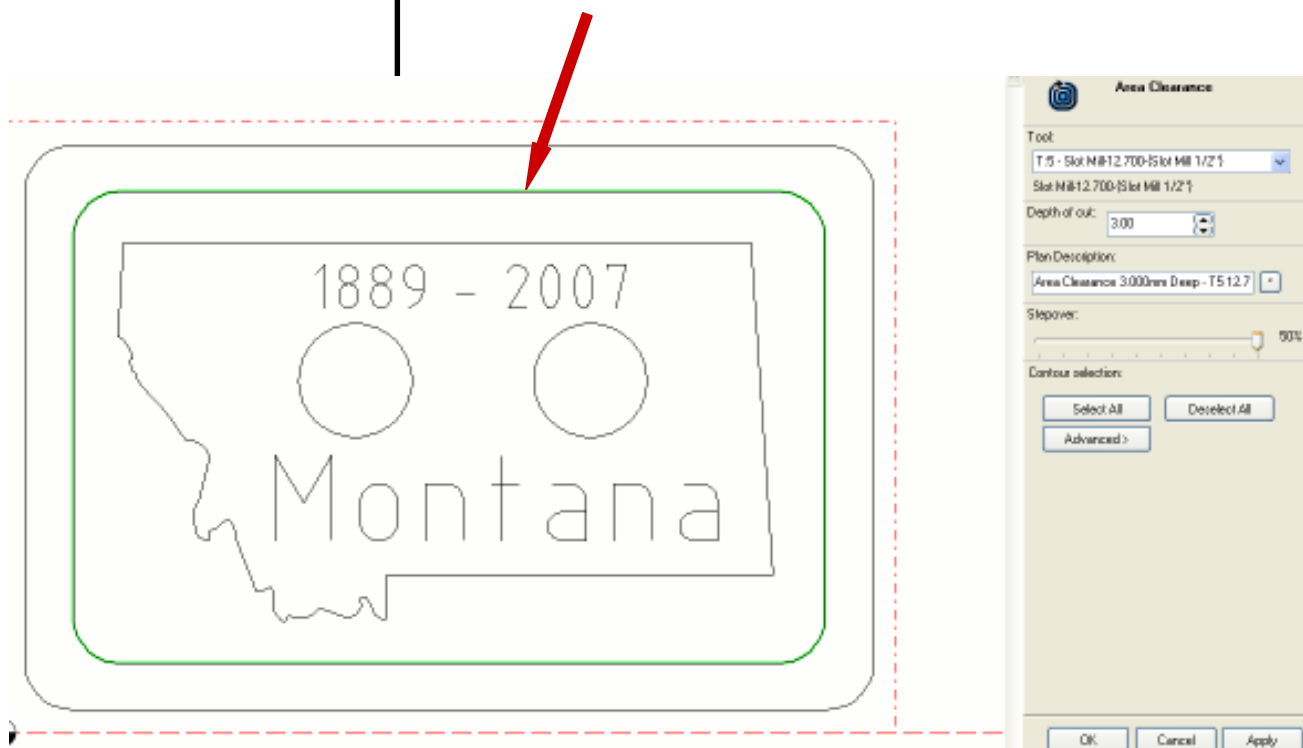
- *Click* on **APPLY**, the *click* **OK**.

10. Add a **Area Clearance Machining Plan**, *click* on Area Clearance.
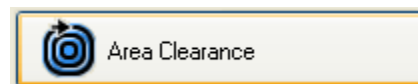
11. Make the following selections …

- Tool: T:5 - **Slot Mill-12.700-{Slot Mill 1/2"}**
- Depth of cut: **4.70**
- Steoover: **50%**

12. Select the **TWO circles** shown below…

- *Click* on **APPLY**, the *click* **OK**.

You are NOT going to create a machining plan for the TEXT at this time. Cutting TEXT uses a different cutting tool. It is easiest to create a separate CNC file late!

13. There should now be FOUR plans in the **Machining Plans window**. (See graphic to the left.)

- If they are all there click NEXT | Next > | !

- If they are NOT all there, see your teacher!

Section 5 - Post Processing

1. Make sure the <u>Machine Type</u> is set to **DENFORD (METRIC) - Milling**.

2. Make sure that your CNC file destination is set to **SAVE** in MY DOCUMENTS\DENFORD\CNC FILES.

   • The file name should use the following format ...
   <Your Initials> State Qrt1 *(example: CBT State Qrt1).*
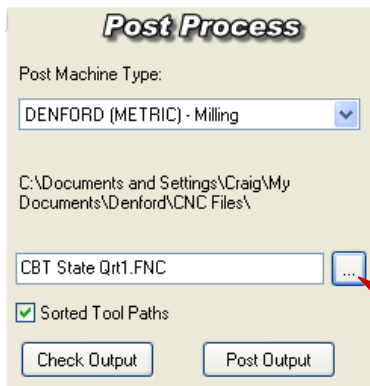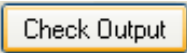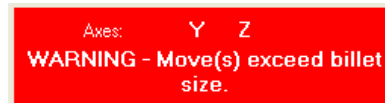
   • *Click* on the BOX located right of the file name.
   (See arrow)

   • Make sure the file name is correct and *click* **OK**.

3. *Click* on **CHECK OUTPUT** to simulate the tool path.

4. A **Red WARNING Window** will appear, this is to warn you that the cutter is going outside of the BILLET size.

   • The purpose of this warning is to let you know that the cutter may run into the clamp(s).

   • This is NOT a problem because you will be holding the material with *double-sided tape* on a **sacrificial base**.

5. Congratulations, you have just created a CNC file that will be machined in VR Milling!
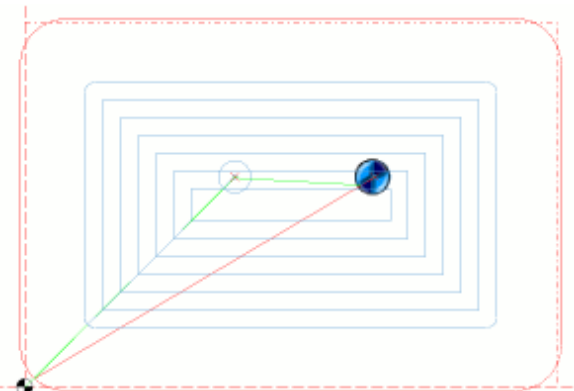
NOW *you are going to create a SECOND CNC file that will machine the TEXT!*

Machining Plans

6. *Click* on the **BACK button** located in the lower right corner of the screen.

7. *Select* the **DRILL machining plan**, then *click* on the **DELETE** button.

8. When the **Confirm Delete menu** appears ... *click* **YES**.

Confirm Deletion

Are you sure you wish delete this cut?

[Yes]   [No]
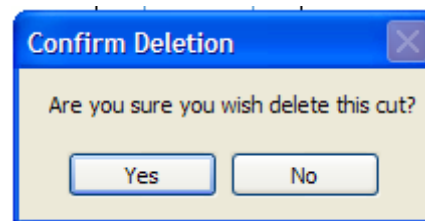
9. Repeat this process until **ALL** machining plans are **DELETED**!

10. You are now ready to create the machining plan for the **TEXT and MONTANA outline**.

11. Add a **Follow Machining Plan**, *click* on Follow.

Follow

12. Make the following selections ...

- Tool: **T:2 - Ball Nose-3.175-{Ball Nose 1/8"}**
- Depth of cut: **.70**

13. Select the objects shown below ... *Click* **APPLY**, then **OK**.

Machining Plans

Follow 0.700mm Deep - T2
3.175mm Diam.

Post Process

Post Machine Type:

DENFORD (METRIC) - Milling

C:\Documents and Settings\Craig\My Documents\Denford\CNC Files\

CBT State Qrt2.FNC

☑ Sorted Tool Paths

Check Output   Post Output

14. There should be <u>ONE plan</u> in the **Machining Plans window**. *(See graphic to the left.)*
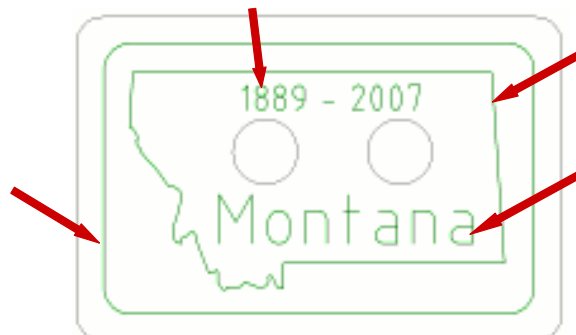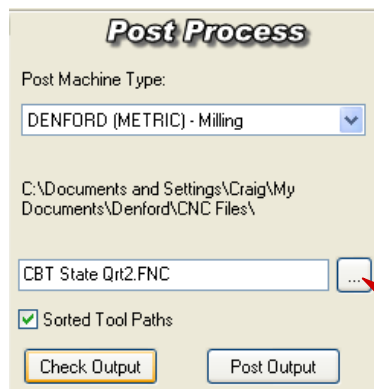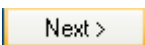
   • If they are NOT all there, see your teacher!

   • If it is there click **NEXT** | Next > | ! and proceed with **post processing**.

15. Make sure the <u>Machine Type</u> is set to **DENFORD (METRIC) - Milling**.

16. Make sure that your CNC file destination is set to **SAVE** in MY DOCUMENTS\DENFORD\CNC FILES.

   • The file name should use the following format ... **<Your Initials> State Qrt2** *(example: CBT State Qrt2).*

   • *Click* on the BOX located right of the file name. *(See arrow)*

   • Make sure the file **name is correc**t and *click* **OK**.

17. *Click* on **CHECK OUTPUT** | Check Output | to simulate the tool path.

18. Congratulations, you have just created a CNC file that will be machined in VR Milling - - - *click* **OK**.
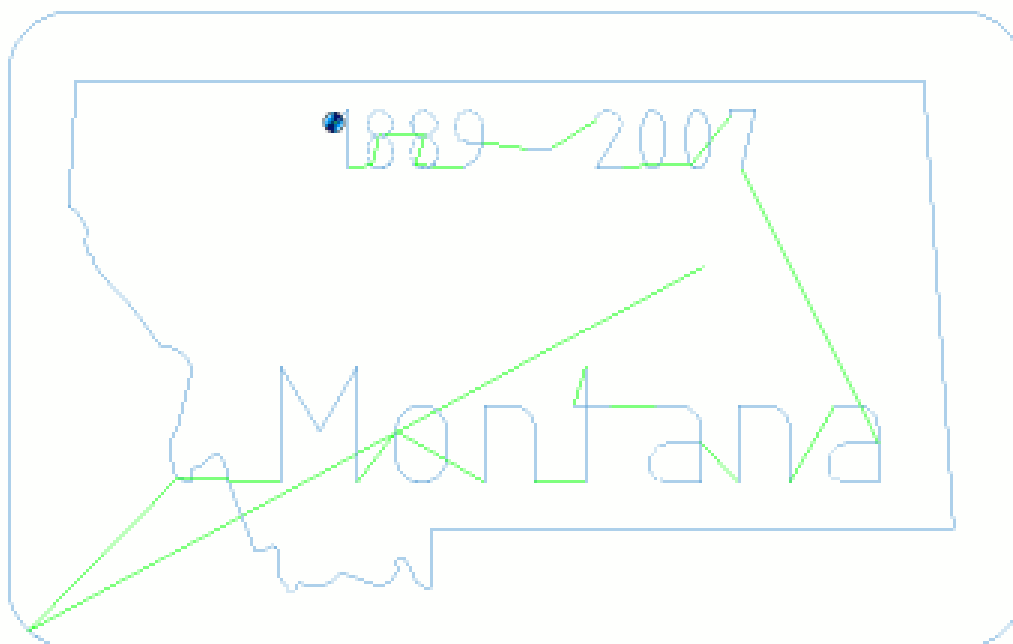
Section 6 - Starting VR Milling

*You will be using a software package called **VR Milling V5** to control a **Denford CNC Router** to manufacture the product you designed earlier in **QuickCAM 2D**.*
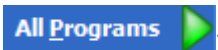
__VR Milling__ is software package that allows full editing and control of CNC (computer numerical control) files, either off-line or on-line.
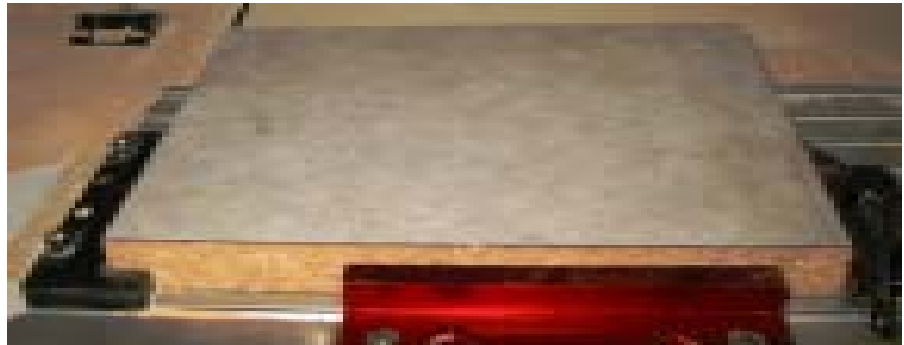
*Before going any further you **MUST** make sure that the Denford CNC Router has a sacrificial base mounted in the router. If this is not done, see your teacher for help! (See example below)*

If <u>VR Milling</u> is not already OPEN follow the steps below, if it is OPEN move to the next page

1. OPEN **VR Milling V5**.

    - *Click* on the **START** menu.

    - Select **All Program**s .

    - Select **Denford**.

    - Select **VR Milling V5**, then **VR Milling V5**
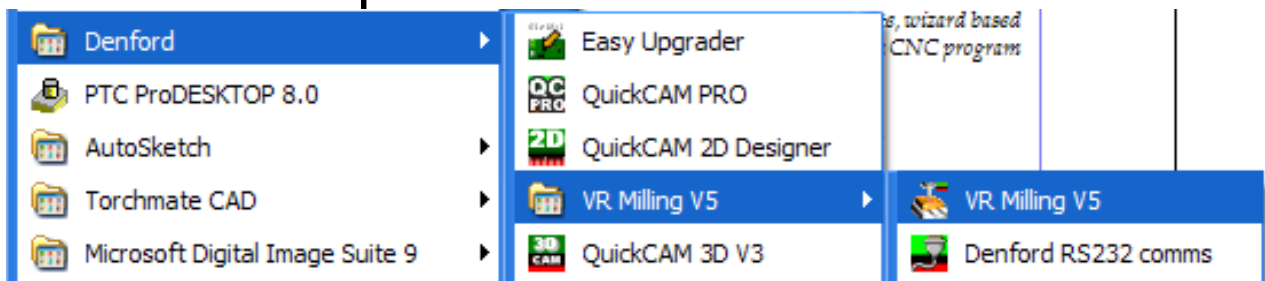
2   Click on the FILE menu at the top left of the screen

- *Select* **OPEN.**

- **Browse** for the file your save earlier in QuickCAM 2D, it should be saved in ...

     MY DOCUMENTS\DENFORD\CNC FILES!

- Remember you saved it as <Your Initials> State Qrt1. *(Example: CBT State Qrt1)*

- The file will have the extension letters **".fnc"**.

- When you have **found** and **selected** the file, *click* **OPEN.**
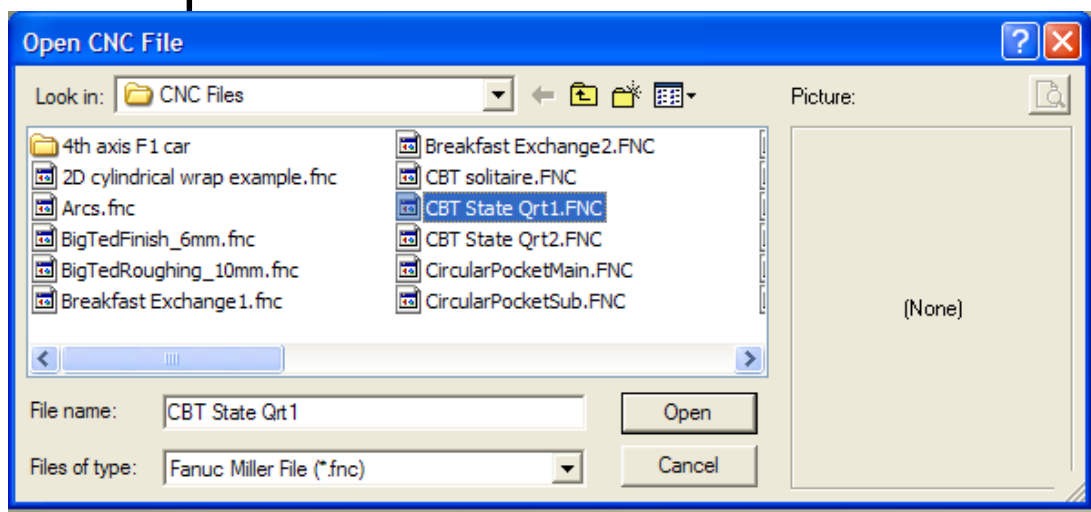
3   The contents of the CNC file will now be displayed in the **EDITOR** window. *See below!*
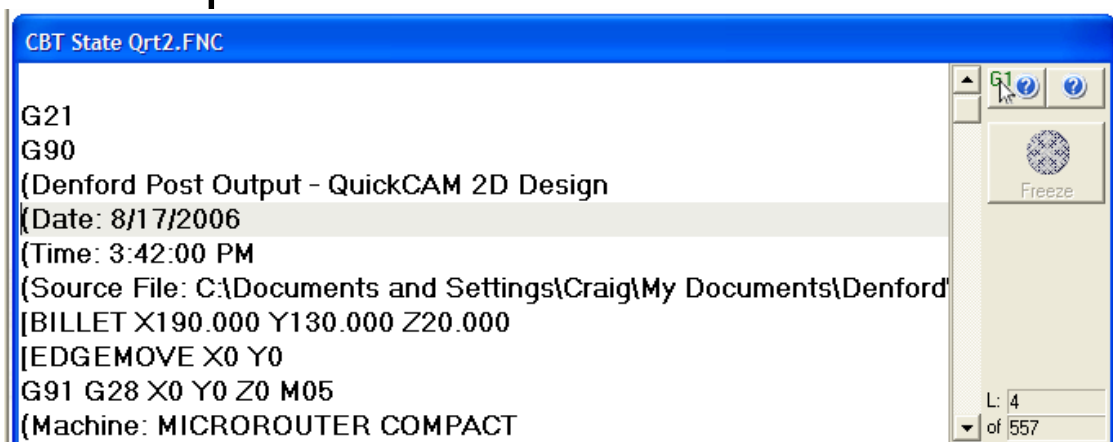
Section 7 - Mounting the Billet

The **"BILLET"** *will be held in place with double-sided tape on a sacrificial base.*

1. Get a **130mm X 190mm billet** from your teacher.

2. Make sure that both faces are smooth - *sand it smooth if necessary!*

3. **Clean** the surfaces of the sacrificial base and the billet.

4. Apply double sided tape to the back of the billet.
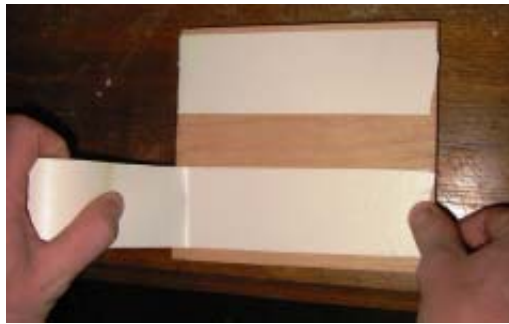
5. Remove the backing from the double sided tape.

6. Place the billet on the **sacrificial base**, taking special care to line up the left and front edges of the billet with the left and front edge of the sacrificial base..

7. Use the **heel of your hand** to apply pressure and vigorously rub back-and-forth to insure the billet is taped down solid.

© Prchal & Todd, 2006

Section 8 - Selecting and Homing the Real Machine

1. From the toolbar on the right side of the screen, select **REAL MACHINE** .

2. Once the connection has been successfully established , the machine **Control Panel** window will appear.
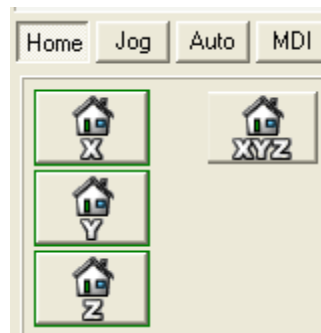
3. Notice, ONLY the **HOME TAB** Home appears.

   • *Click* on the **Home All** XYZ button to home all three machine axis.

   • After HOMING, the **Jog**, **Auto** and **MDI** tabs become available. *See below!*

Section 9 - Move the Machine Head and Fit the Cutting Tool

1. In the **control panel**, *click* on the **JOG** Jog tab to select Jog mode.

2. Click on the **Jog button** to change between **Continuous** and **Incremental modes.** *Try it!*

   • To change the position of the cutting tool **quickly**, use Jog Continuous mode.

   • To change the position of the cutting tool **incrementally**, use Jog Incremental mode.

3. Notice how the **number changes** below the Feed Control Knob when you switch between the two modes!

Jog Feed Control Knob

This is next step is very important for you to understand, after reading through the information below, see your teacher for a quick review!

4. Feed Rate is controlled by the **Jog Feed Control Knob**.

   • The value is shown in the **readout** below the control knob.

   • To change the **Feed Rate**, *click on and drag* the Jog Feed Control Knob **up** and **down**.

   • *Watch the feed rate values change as you move the mouse up or down!*

5. The **four cursor (arrow)** keys and the **Page Up** and **Page Down** keys on the keyboard are used to control the **X**, **Y** and **Z** axis.

   • ← moves the tool **LEFT** on the X axis.
   • → moves the tool **RIGHT** on the **X** axis.
   • ↑ moves the tool **BACKWARD** on the **Y** axis.
   • ↓ moves the tool **FORWARD** on the **Y** axis.
   • **Page Up** moves the tool **UP** on the **Z** axis.
   • **Page Down** moves the tool **DOWN** on the **Y** axis.

♦ Take 5 minutes to practice jogging the tool around the work area in **Continuous** and **Incremental** modes.

♦ Also practice changing the **jog feed rate**.

♦ It is important that you feel comfortable with these controls before moving on!

Section 10 - Selecting the Work Offsets

Offsets are the distances the cutting tool needs to travel, from it's HOME position to the point from which the program starts in X, Y and Z.
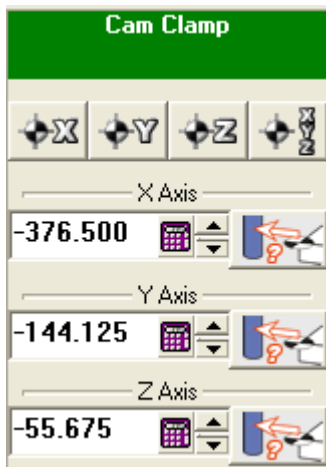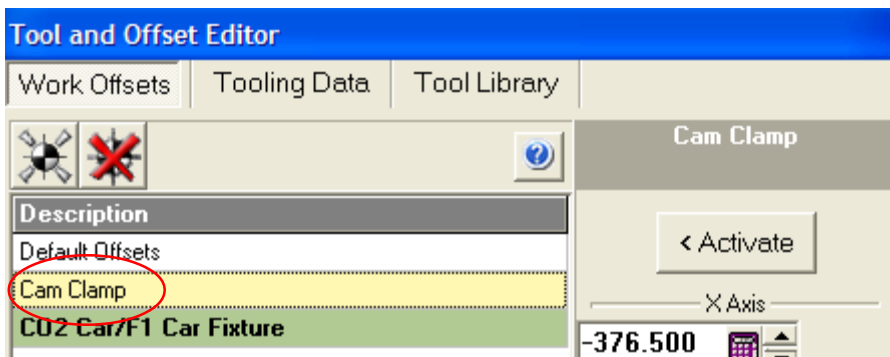
1. *Click* on the **Tool and Offset Editor** button located at the bottom left of the screen

2. *Click* on the **Tooling Data tab** - *select* **Slot Mill 1/2**".
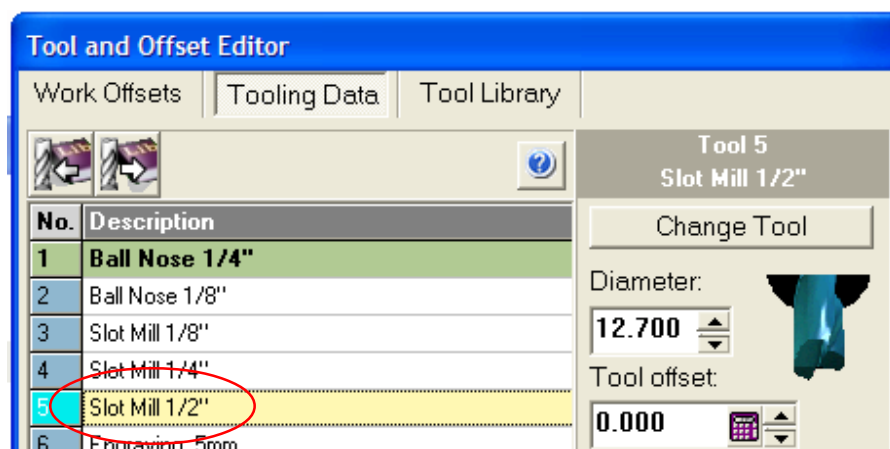
3. *Click* on the **Work Offsets tab**, select the **Cam Clamp**, then *click* ACTIVATE.

4. The window to the left will appear, the **X** and **Y** offsets have already been set for the CAM CLAMP.

5. You will set the **Z offset** using what you learned about jogging the cutter head in **Section 9.**

6. Move the cutter over the billet, jog the cutter downward until it is just above the billet.

If you are having trouble ... see your teacher for help!

7. Open the guard, and place a sheet of paper under the cutter.

8. Move the cutter back and forth until you feel the paper being grabbed by the cutter, this is the correct height.
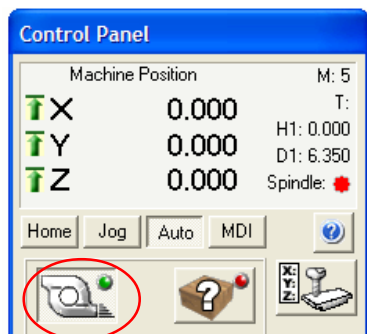
9. Set the datum by *clicking* on the **Set Datum for Z Axis button**.
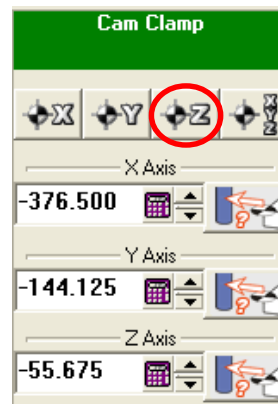
10. CLOSE the **Tool and Offset Editor** by *clicking* on the **Tool and Offset Editor** button located at the bottom of the screen.

*Tool is to CLOSE to billet, Jog the tool **up** to clear the billet!*

Section 11 - Run the Program

1. Before **RUNNING** the program HOME ALL the axis, and make sure the cutting tool is clear of the billet.

    • If NOT, use the **cursor**, **page up** and **page down** keys to move the cutter clear of the billet!

2. Select the **HOME TAB** Home in the control panel.

3. *Click* on **HOME ALL** , wait for the homing process to finish.

4. Click on the AUTO TAB Auto .

5. Select TURBO MODE , make sure the little light is green and not red.

    • This will greatly reduce large 3D object machining times.

    • It will usually make the machine perform with a smoother motion.

6. The program is now ready to **RUN**.

    • Do you have a billet installed?

    • Is the 1/2" Slot Mill installed?

7. If ALL setups are **OK, go** to the **File Control** toolbar located at the bottom of the screen.

Play   *Stop*

Pause   *Rewind*
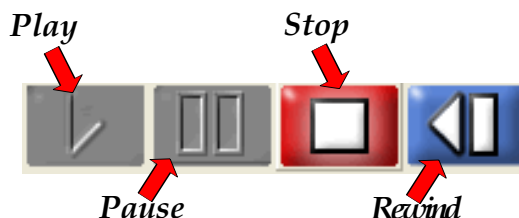
8. *Click* on the **STOP** 🔲 button, followed by the **REWIND** ◀❙❙ button.

   • This will ensure that the program starts from the beginning.

   • **DO NOT** *click* **PLAY** until you read through **Section 12 - Monitoring the Machining Process.**

9. *Click* on the **PLAY** ▶ button to start the program.

   The program will begin to run.

10. A message will appear asking you to change to tool number 5, *You already have the 1/2" Slot Mill installed.*

Section 12 - Monitoring the Machining Process

1. It's time to set back, LOOK and LISTEN ...

   • Keep your hands by the red **EMERGENCY STOP** button at all times.

   • Make sure that the tool does not run into the Cam Clamp.

   • Listen to the *sound of the machine*, if the cutter is laboring, you may want to use **feed rate override knob** on the front of the machine.

   • If you see extreme vibration you make want to reduce the feed rate.

> *TIP:*
> ♦ *To gain more control you may want to use the FEED RATE OVERRIDE KNOB on the front of the router to reduce the feed rate when first starting a program.*
>
> ♦ *After you are happy with the way to tool is cutting you may increase the Feed Rate to 100%.*

2. You are now ready to begin machining!

    • *Click* **OK**.

    • The **spindle will start**, and the program will begin to run!

    • PAY CLOSE ATTENTION until the machining STOPS!

3. When the machining is DONE <u>**do not remove**</u> the **billet** from the machine!

Photo of machined product!

4. You are now going to load the **second CNC file** you created - - <u>*Remember*</u> you saved it as ...
    <Your Initials> State Qrt2. *(Example: CBT State Qrt2)*

5. *Click* on the **FILE menu** at the top left of the screen

    • *Select* **OPEN.**

    • **Browse** for the file your save earlier in QuickCAM 2D, it should be saved in ...
        MY DOCUMENTS\DENFORD\CNC FILES!

    • When you have **found** and **selected** the file, *click* **OPEN.**

6. *Click* on the **Tool and Offset Editor** button located at the bottom left of the screen
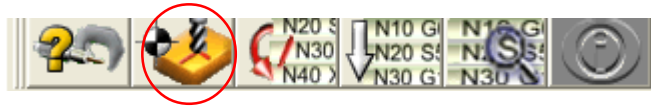
7. *Click* on the **Tooling Data tab**.

8. When the window below appears - *select* **Ball Nose 1/8**".
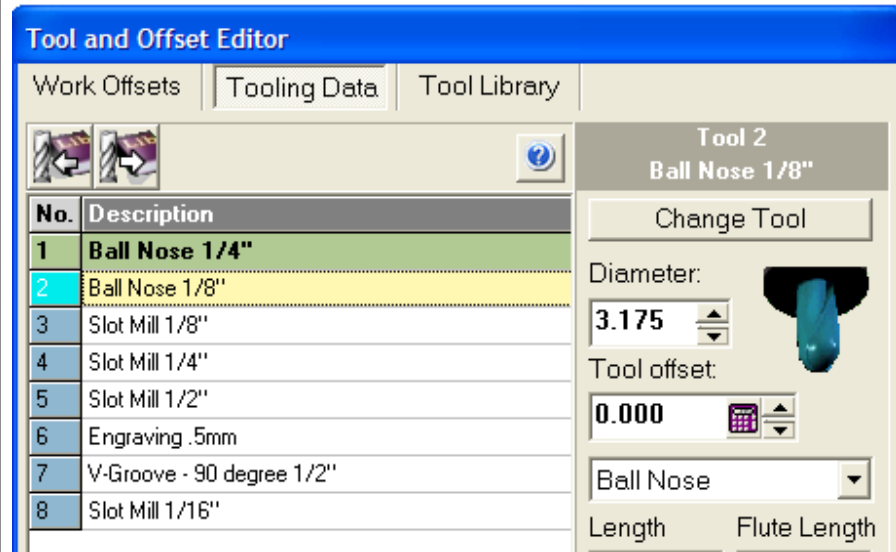
9. Make sure the **1/8" Ball End** is installed in the router!

10. *Click* on the **Work Offsets tab**, and use the techniques you learned earlier to touch on to the top of the billet.

11. Note: When you touch on top of the billet, make sure you are touching on the pocketed area! *(see arrow below)*

*Make sure that you are TOUCHING on the **Pocketed Area** when setting up the 1/8" Ball End Mill.*

12. Follow the procedures that you learned earlier in this activity to **RUN the program**.

13. **Start the program** - - set back, LOOK and LISTEN ...

- Keep your hands by the red **EMERGENCY STOP** button at all times.

- Make sure that the tool does not run into the Cam Clamp.

- Listen to the *sound of the machine,* if the cutter is laboring, you may want to use **feed rate override knob** on the front of the machine.

- If you see extreme vibration you make want to reduce the feed rate.

14. When finished machining - - show the completed product to your teacher!