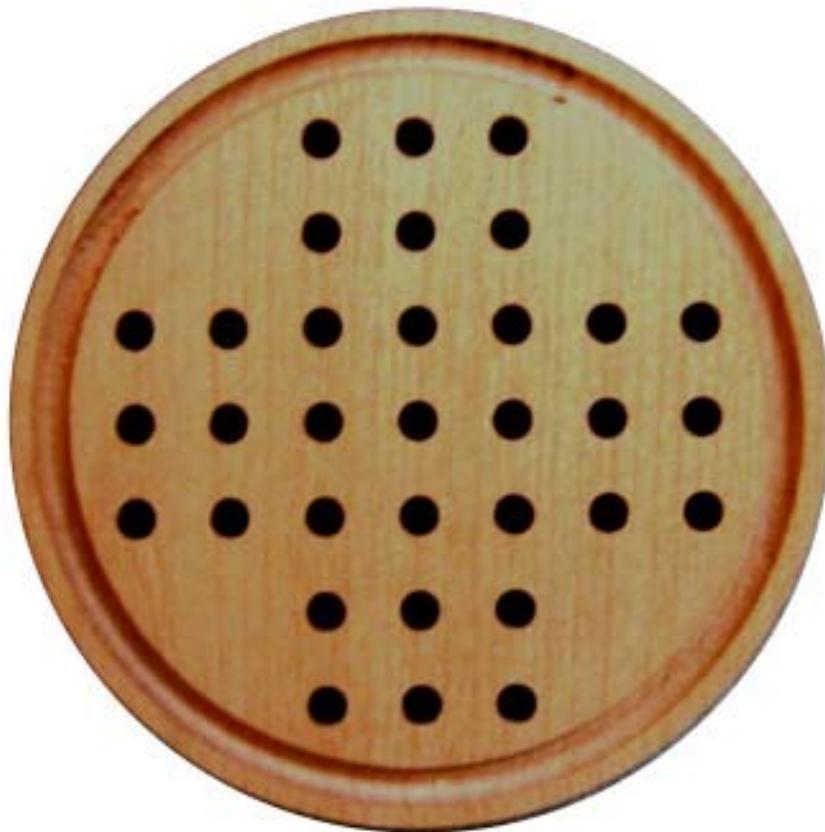


SOLITAIRE PEG GAME



Section 1 - Starting the QuickCAM 2D Software

1. OPEN QuickCAM 2D.

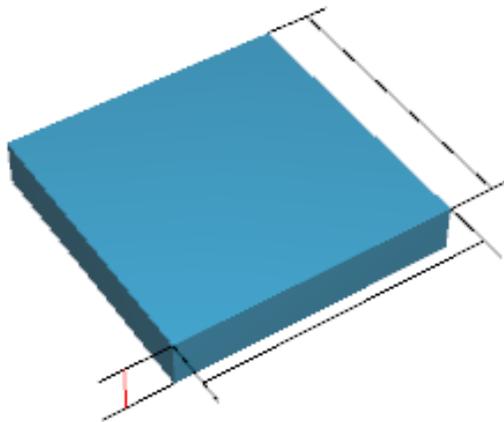
- Click on the **START**  menu.
- Select **All Programs** .
- Select **Denford**.
- Finally select **QuickCAM 2D Designer**.

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*.



Width - X	<input type="text" value="140"/>
Depth - Y	<input type="text" value="140"/>
<input type="button" value="Match to Printer..."/>	
Height - Z	<input type="text" value="20"/>

Section 2 - Setting up the Drawing Parameters



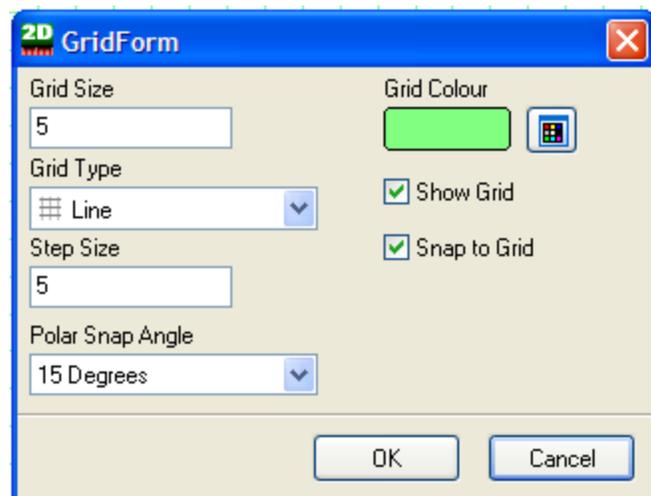
1. Go to the **OPTIONS menu** at the top of the screen.

- Select **Units**.
- Select **Metric**.

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

- Select **GRID**  **Grid...**

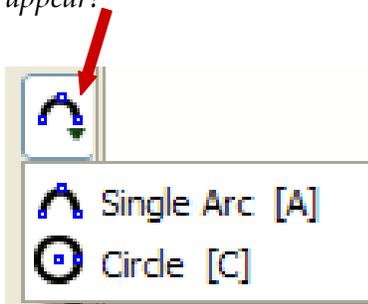
3. When the **Grid Form** appears, make the following changes ..



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

4. You are now ready to start designing!

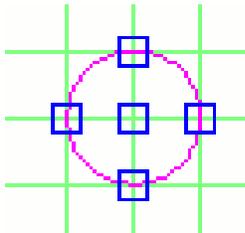
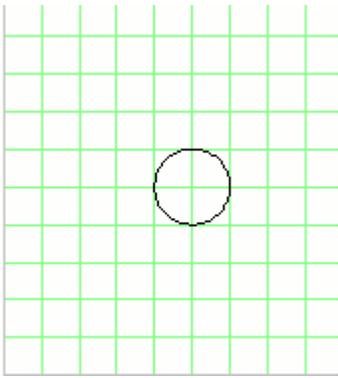
Click and hold to make the DROP-DOWN menu appear!



Section 3 - Creating Your Design

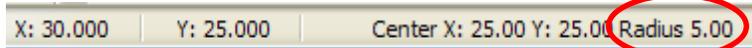
1. Select the **CIRCLE tool**  from the toolbar on the left side of the screen.

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



2. Move the cursor until the coordinates at the bottom of the screen read ... X: 25.000 Y: 25.000

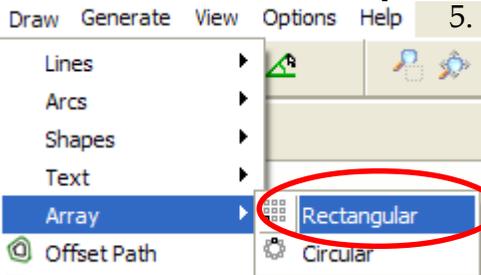
3. Click and drag the cursor to the right until you have a 5mm radius.



- Click the mouse button and a circle will appear.

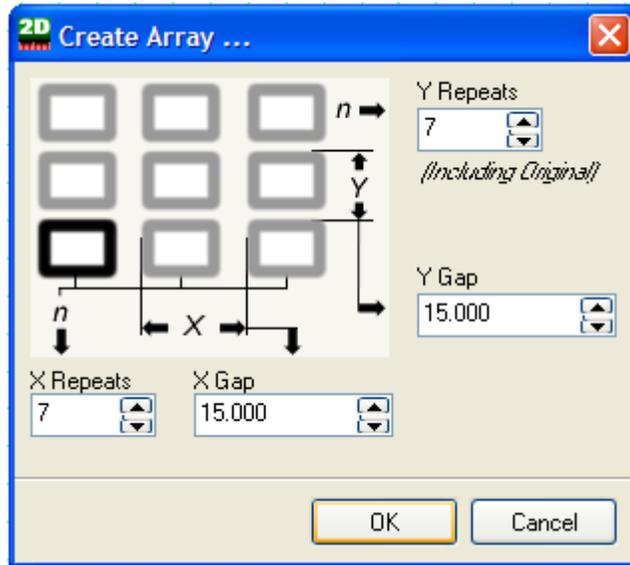
4. Choose the **SELECT tool**  from the toolbar on the left.

- Click on the circle you just created.
- After clicking on the circle you will notice a change in its appearance. See to the left.
- This indicates the object is now selected and that it can be edited.

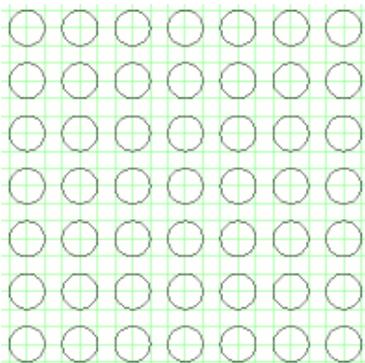


5. From the top menu select **Draw**, then **Array** and finally **Rectangular**.

- When the **Create Array window** opens make the following changes ...

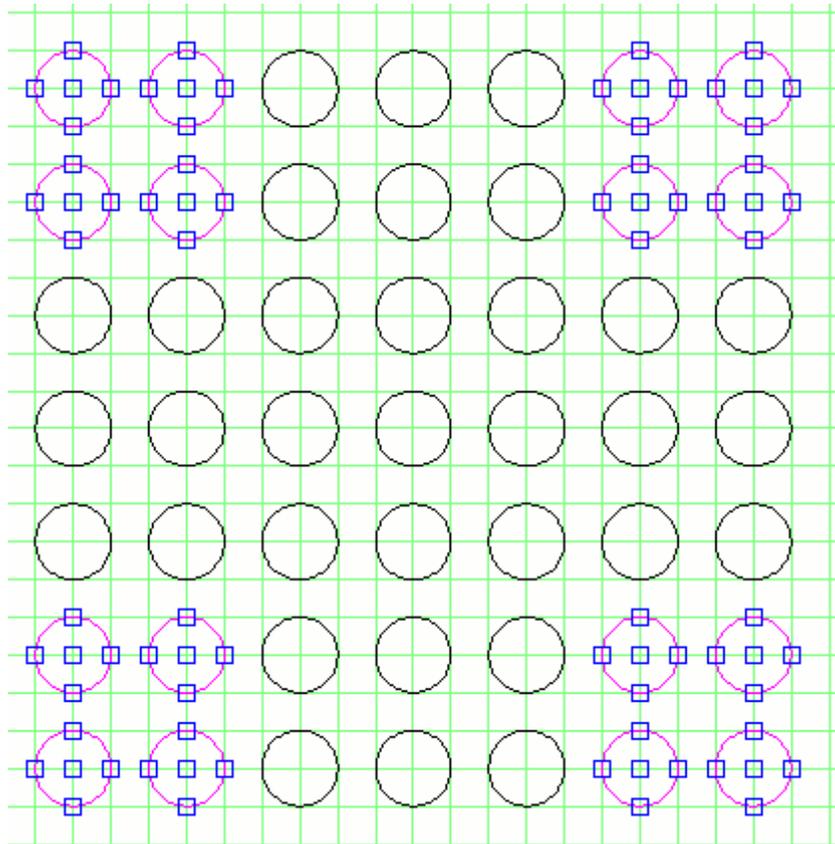


- After the changes have been made, click **OK**.



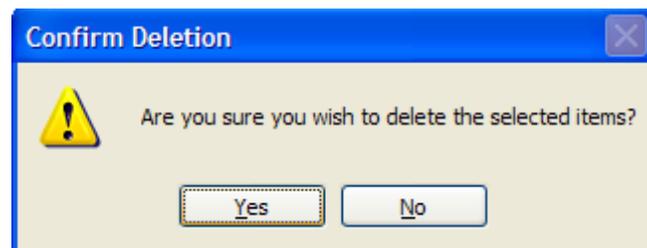
6. You will now select and delete 16 of the circles that you just created!

- Chose the **SELECT tool**  from the toolbar on the left.
- *Click* on the circle located in the lower left
- While *holding down the shift key*, continue to select the other 15 circles shown below.



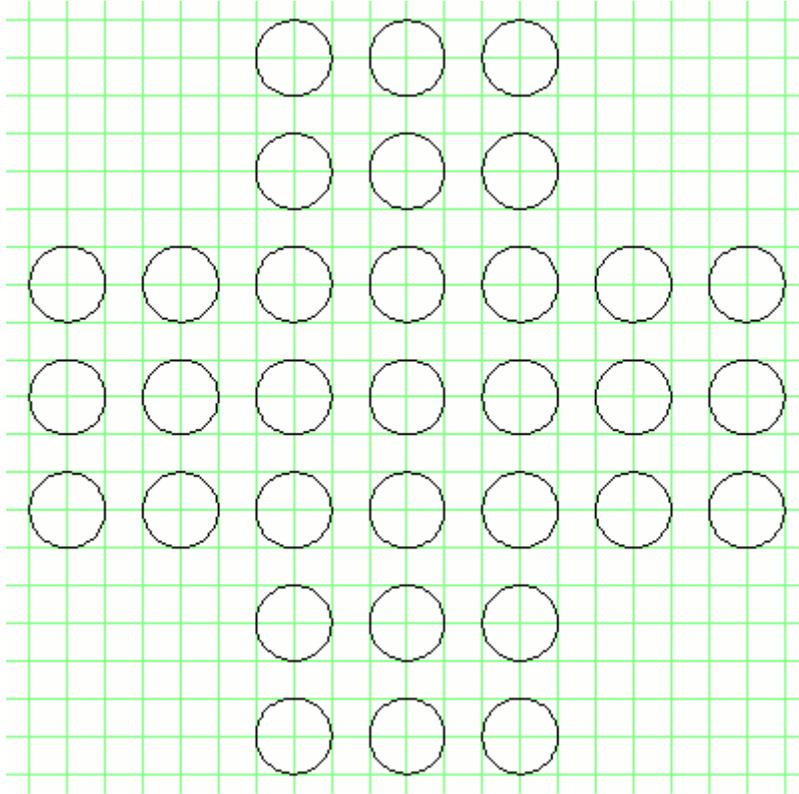
- When you have selected the circles as shown above, press the **DELETE** key on the keyboard.

7. When the window below appears, *click* on **YES**.



8. The image on the screen should now look like the image below.

- If it doesn't see your teacher now!

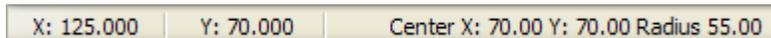


9. Select the **CIRCLE tool**  from the toolbar on the left side of the screen.

10. Move the cursor until the coordinates at the bottom of the screen read ...

X: 70.000	Y: 70.000
-----------	-----------

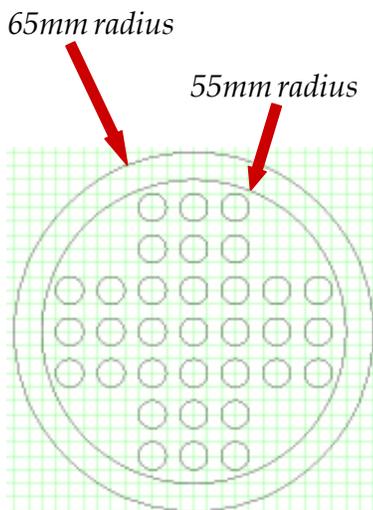
11. *Click and drag* the cursor **to the right** until you have a **55mm radius**.



- Then *click* to mouse button, a circle will appear.

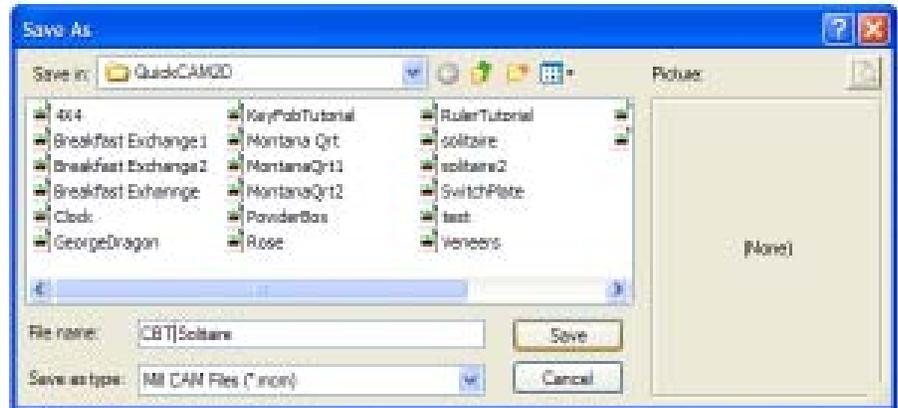
12. Repeat the above process to create a **65mm circle**.

- The design should now look like the *image on the left!*



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> Solitaire (example: *CBT Solitaire*).

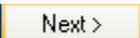


14. Show the **finished design** to your teacher!

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**.

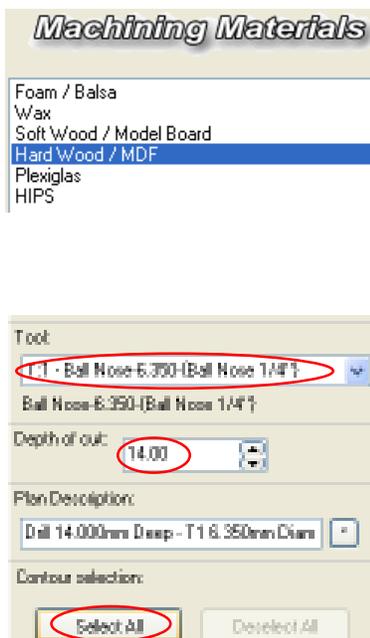
- Click **NEXT**  to continue on!

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



3. Make the following selections ...

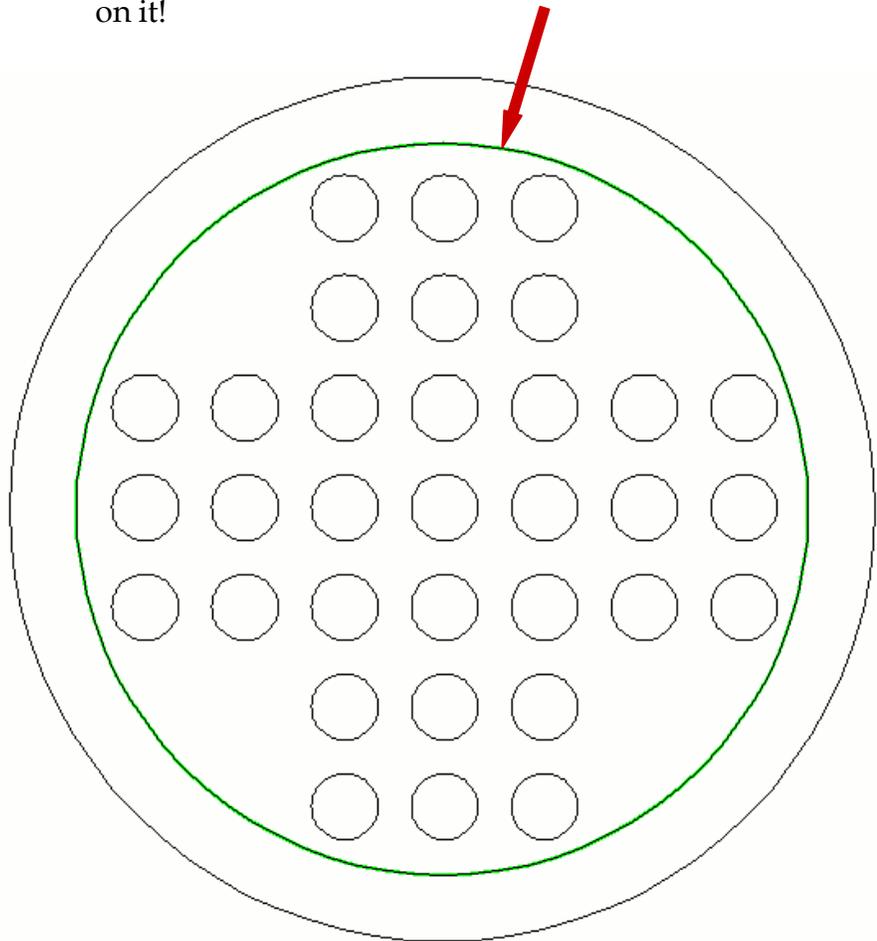
- Tool: T:1 - Ball Nose-6.350-{Ball Nose 1/4"}
- Depth of cut: 14.00
- Contour Selection: **Select ALL**



4. Click on **APPLY**, then *click OK*.
5. Add a **Outside Offset Machining Plan**, *click* on Outside Offset.



6. Make the following selections ...
 - Tool: T:1 - **Ball Nose-6.350-{Ball Nose 1/4"}**
 - Depth of cut: **3.20**
7. 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*.

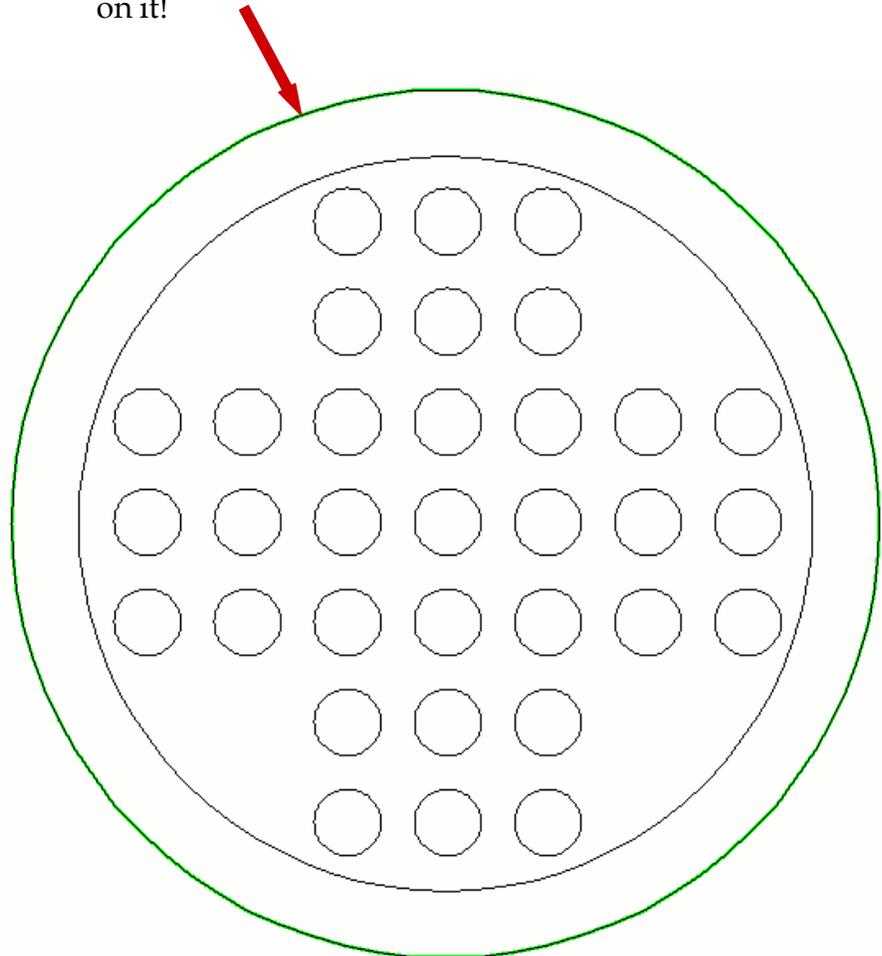
8. Click on Outside Offset.



9. Make the following selections ...

- Tool: T:1 - Ball Nose-6.350-{Ball Nose 1/4"}
- Depth of cut: 23.20

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

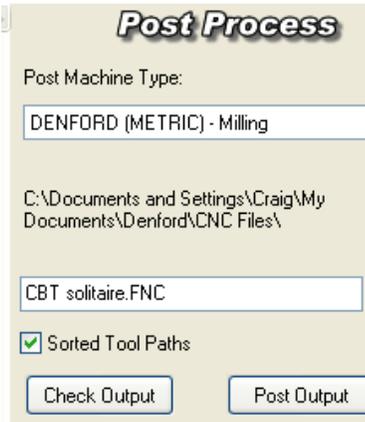


- Click on **APPLY**, then click **OK**

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



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



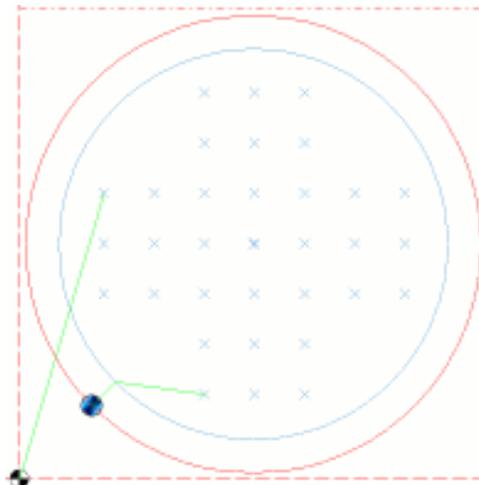
Section 5 - Post Processing

1. Make sure the Machine Type 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> Solitaire** (example: *CBT Solitaire*).
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. Click **OK**.
 6. Congratulations, you have just created a CNC file that will be machined in VR Milling!



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**.

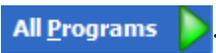
VR Milling is software package that allows full editing and control of CNC (computer numerical control) files, either off-line or on-line.

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 **VR Milling** 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 Programs .
- Select Denford.
- Select VR Milling V5, then VR Milling V5



2 Click on the FILE menu at the top left of the screen

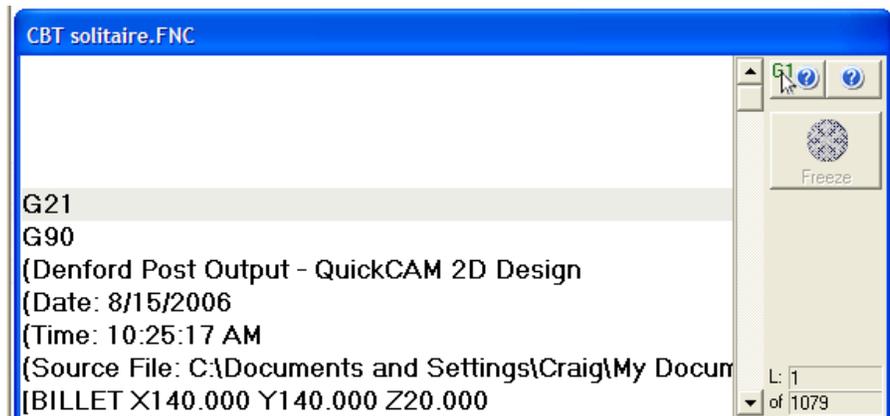
- *Select* **OPEN**.
- **Browse** for the file you save earlier in QuickCAM 2D, it should be saved in ...

MY DOCUMENTS\DENFORD\CNC FILES!

- Remember you saved it as <Your Initials> Solitaire.
(Example: CBT Solitaire)
- The file will have the extension letters "**.fnc**".
- When you have **found** and **selected** the file, *click* **OPEN**.



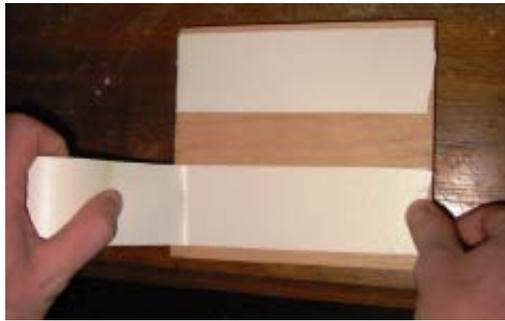
3 The contents of the CNC file will now be displayed in the **EDITOR** window. *See below!*



Section 7 - Mounting the Billet

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

1. Get a **140mm X 140mm 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.



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**  appears.

- Click on the **Home All**  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**  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!

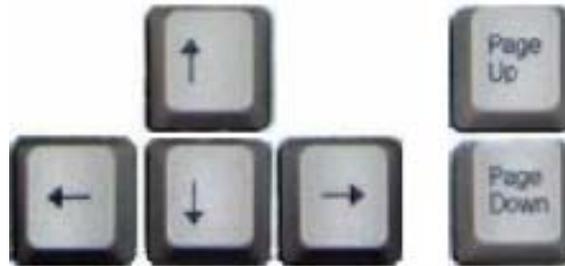
4. Feed Rate is controlled by the **Jog Feed Control Knob**.

- 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!

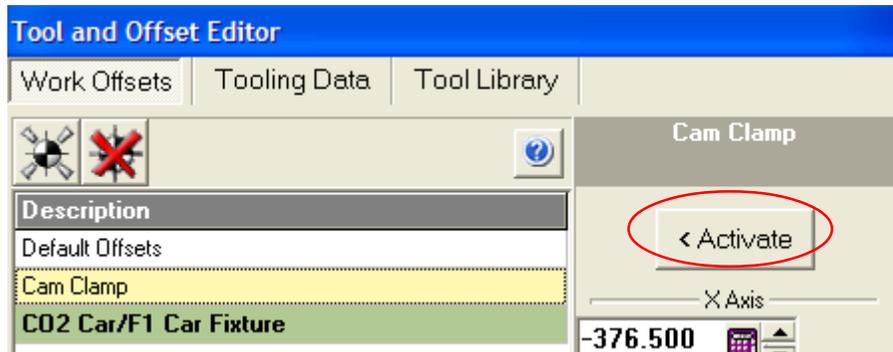
Section 10 - Selecting the Work Offsets

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.

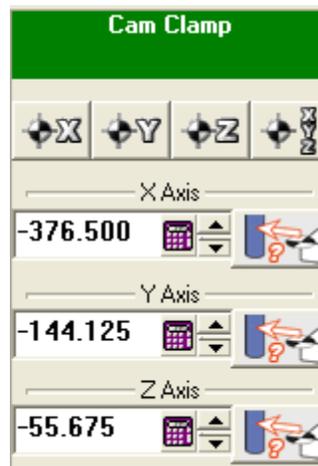
1. Click on the **Tool and Offset Editor** button located at the bottom left of the screen



2. When the window below appears, select the **Cam Clamp**, then click **ACTIVATE**.



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



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

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

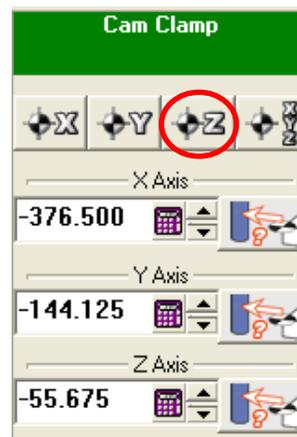
5. Move the cutter over the billet, jog the cutter downward until it is just above the billet.



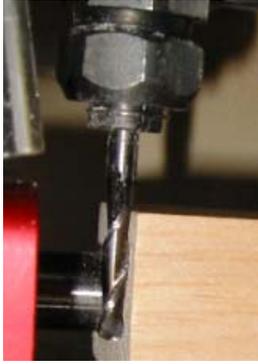
6. Open the guard, and place a sheet of paper under the cutter.
7. Move the cutter back and forth until you feel the paper being grabbed by the cutter, this is the correct height.



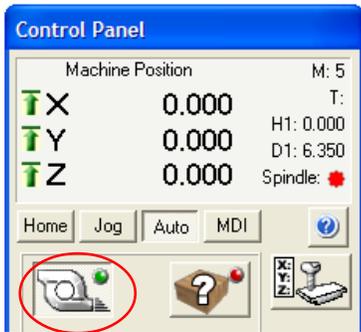
8. Set the datum by *clicking* on the **Set Datum for Z Axis button**.



9. CLOSE the **Tool and Offset Editor** by *clicking* on the **Tool and Offset Editor** button located at the bottom of the screen.

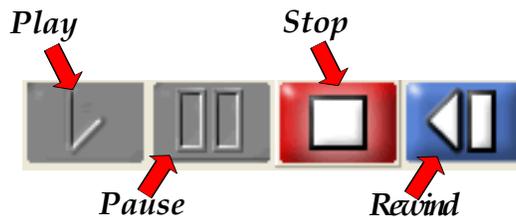


Tool is too CLOSE to billet,
Jog the tool **up** to clear the
billet!



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**  in the control panel.
3. Click on **HOME ALL** , wait for the homing process to finish.
4. Click on the **AUTO TAB** .
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 1/4" Ball End mill installed?
7. If ALL setups are **OK**, go to the **File Control** toolbar located at the bottom of the screen.



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 1, *you should have already installed the 1/4" Ball End mill.*



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!

Enjoy the Show!!

