

# MasterCAM 9.1 – 3D MODEL CREATION

Source = <http://www.gsd.harvard.edu/inside/cadcam/mastercam/>

## This tutorial will assist in:

1. Importing a 3D Model into MasterCAM
2. Prepare the model for creating a usable tool path
3. Set machining parameters
4. Design a tool path (for both rough and finish cuts)
5. Export the tool path g-code as a machine ready \*.nc file

## BACKGROUND:

MasterCAM is a powerful software able to both design 3-D models and prepare files for machining. This tutorial will focus exclusively on the latter capabilities. More specifically, this tutorial will enable you to create a machine-ready file (for example, a \*.nc file) from an imported 3-D model. The file this tutorial will help you create will be usable with a variety of g-code based machines including the TCM1000 mill and AXYZ4008 router here at the GSD. To use this tutorial, you will need a 3-D model that you have saved using any one of the following formats:

**STL**  
**VDA**  
**SAT**  
**ProEAscii**

**STEP**  
**Autodesk**  
**IGES**  
**Parasld**

Note: Mastercam can read both surfaces and solid models, but they must be originally drawn in environments that produce "watertight" models. FORM Z IS NOT SUITED TO PRODUCING THESE MODELS.

Note: We have had very good luck using an .igs or .iges file format in particular, and recommend that you use this format unless you have a compelling reason to use a different type.

# Import Your Model

1. Open MasterCam Mill (Icon: Mill 9.1)
2. Using the prompts at the top left of the screen, proceed through these clicks: **File/Converters/<Click your file's converter type>/Read File**
3. Browse to find your file and open it
4. A dialogue box will appear, accept defaults and click **OK** and your model should appear on screen.
5. If a warning asks if you would like to delete the current tool, click, **No**. Your model should now appear.
6. Position your model in the window by using the fit, zoom, and rotation buttons.

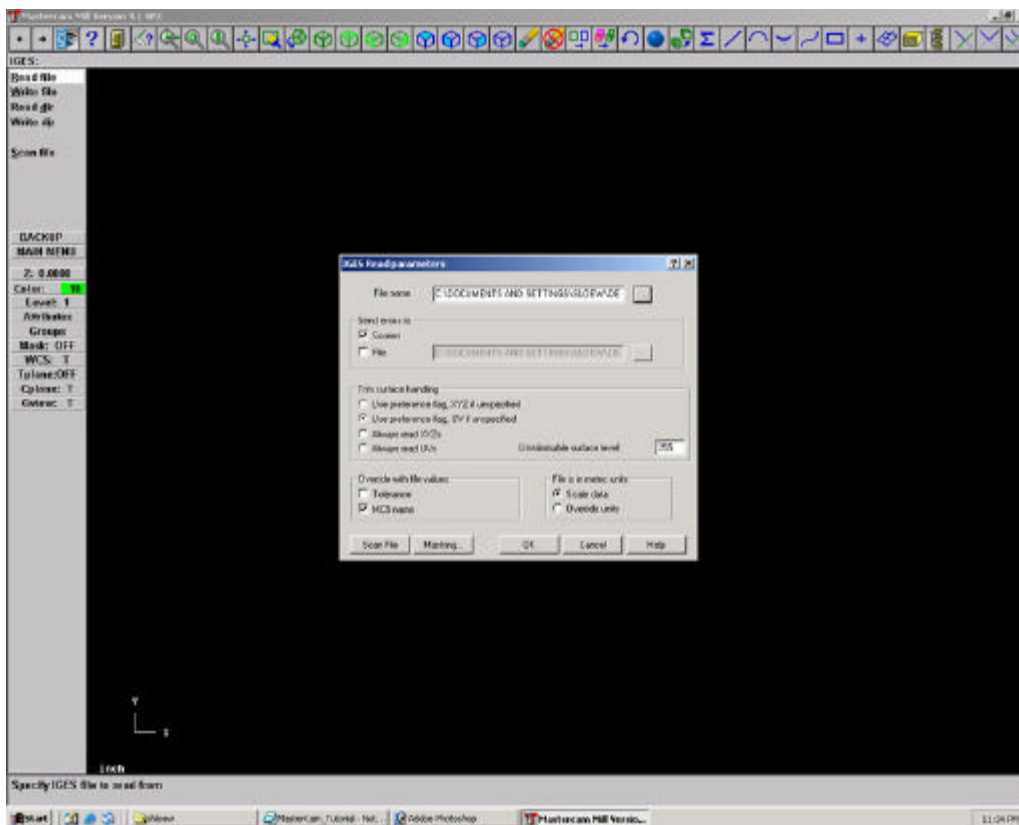


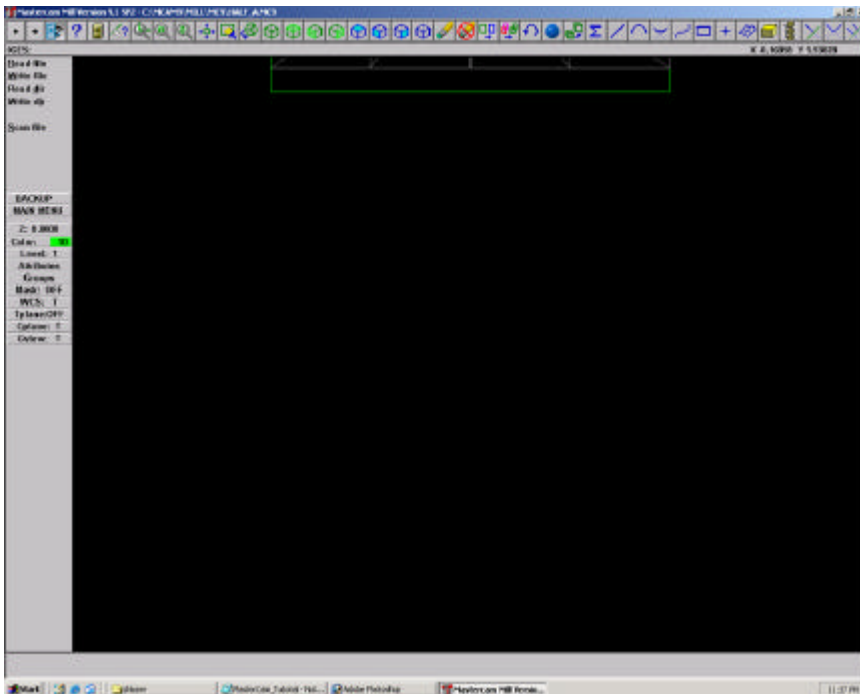
Figure 1: Import your model: IGES "read parameters" dialogue box appears when opening an IGES file. Click OK, accepting all defaults.

## Clean Your Model

(this step is usually optional if you have created a good model or surface in the original program)

Sometimes an import may have some odd, residual lines that are not part of your design or that you do not want to be read by the machine. You want to delete these by clicking on the delete tool and selecting each line for deletion. You may need to tumble your object to get to all of these lines.

1. The "**Screen-Repaint**" button will refresh the image on the screen once you are done.



**Figure 2: Import your model: when first importing, your model may appear just off screen. Use the screen view buttons to adjust your view.**

2. You may also use the string of clicks to delete all non-surfaces:

**Main Menu/Delete/All/Lines...Arcs...Splines...Points...**, etc.

When done, click *Main Menu* to exit from the delete function. You may also need to click the **Screen-Repaint** button to refresh the image on the screen.

3. It is possible that some of your object may be a different color than the rest. This may be suggesting potential problem areas for the machining. To verify the viability of the surfaces, follow this click sequence. This will first test the normals:

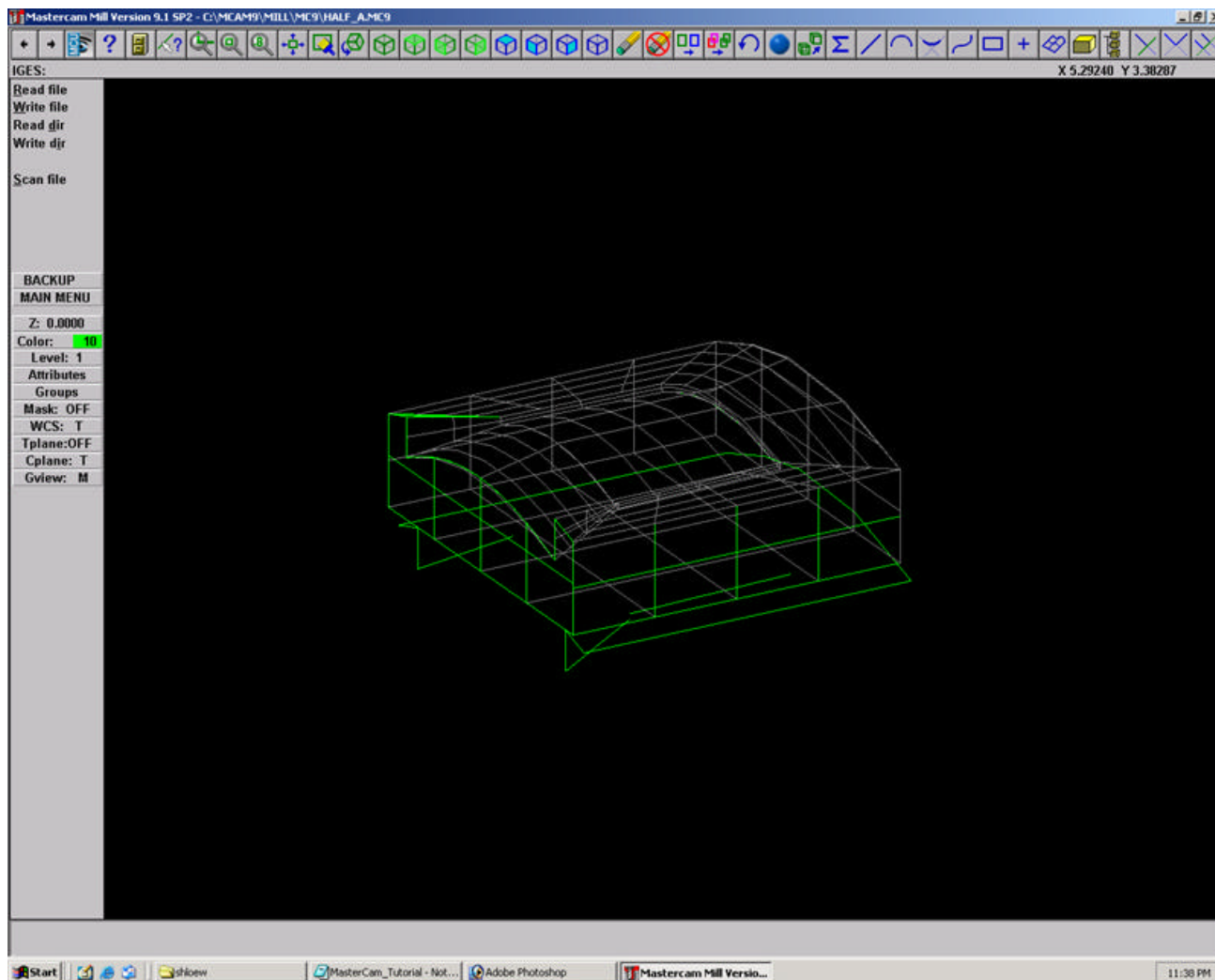
**Main Menu/Analyze/Surfaces/Test Norms/All/Surfaces/Done**

A pop-up window will then appear informing you of your model's integrity. If you have reversed normals, follow the on screen prompts to adjust the surfaces or rebuild your model and re-import.

This next sequence will check the model for sharp internal corners that may not be machinable:

**Main Menu/Analyse/Surfaces/Check Model/All/Surfaces/Done**

Then type enter to accept the tolerance shown at the bottom left of the screen.



**Figure 3: Clean your model: Step 3. You may need to delete excess lines, points, arcs, etc. from your model. You want only surfaces when creating tool paths.**

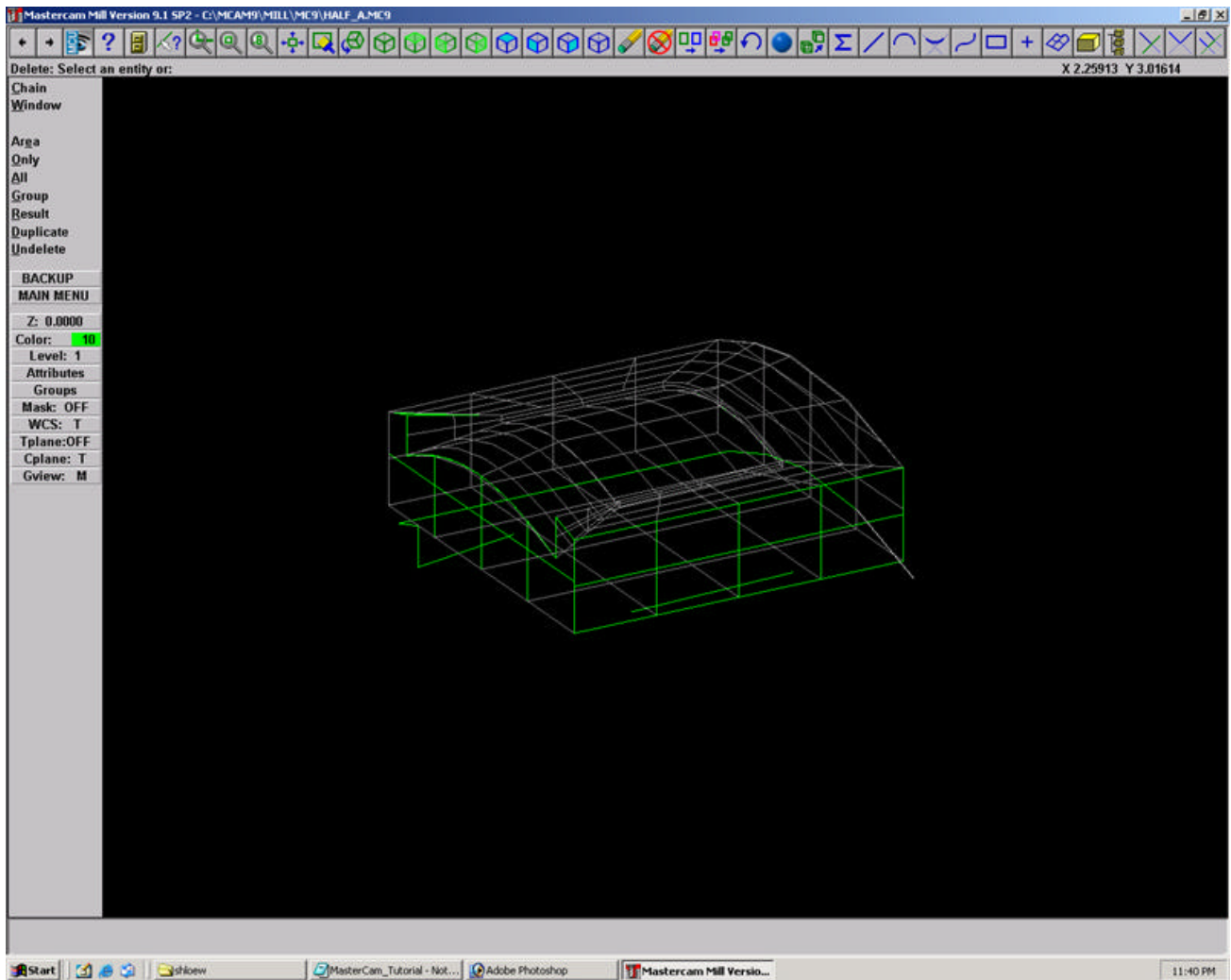


Figure 4: Clean your model: Step 3. Notice all excess lines, arcs and splines have been deleted.

A pop-up window will appear with diagnostics of your model. Click **OK**. If you had internal sharp corners, MasterCam will ask you if you wish to draw the internal sharp curves. We recommend that you say no and either proceed knowing that the machine may not be able to reproduce your model as precisely as you have drawn it or to redraw your model in the original modeling program avoiding sharp internal corners and re-import. Take note of the location of the curves that MasterCam indicates contain sharp internal corners before proceeding. And also understand that you may be able to set parameters that will minimize the differential between what is modeled and what the machine is capable of cutting. See below for details.

## Position Your Model and Set Origin (this is an important first step)

Once you have cleaned up your file you need to set the origin in MasterCam. Press F9 to toggle on and off the X,Y, and Z axes. Use the following steps to ensure that your model is in the positive XY corner of the MasterCam modeling space and that your machining surface is facing upward in the positive Z direction.

### **To rotate your model:**

1. To rotate your model around the Z axis work in Isometric View. To rotate your model around the X/Y axis work in Side view. Then click thru as follows:

**XFORM/Rotate/All/Entities/<select your piece>/Done**

2. Click *Relative* and select the corner of your piece around which you would like to rotate it.

3. Then click on **Polar**. You will be prompted in the bottom left hand corner to enter a relative distance. Keep the default of 1 and then enter the angle or rotation that you desire. The 'Rotate' window will pop-up and you will then be prompted to specify an angle. Make sure that Move is selected and that the 'number of steps' equals 1. (Point and Sketch can also be used for rotating. Experiment to find what works for your design.)

Once your model is facing up properly you can move it to line up with Mastercam's origin. If your model does not have a point that can be brought to the origin while keeping the whole piece with the positive XY quadrant, just move it close to Mastercam's origin.

### **To set Origin:**

1. Move the 0,0,0 corner of your model to the positive XY corner of Mastercam. To do so, view the modelspace in Isometric View and click...

**XFORM/Translate/All/Entities/DONE**

Your entire model should be highlighted.

2. To move your model to the origin click "**Between Pts**". A prompt will come up in the bottom left hand corner of your screen asking you to select the point to move from. Select the corner of your model that you would like to move to the origin. Then select the origin as your destination point. The translate dialogue box will appear and ask you if you want to move, join, or copy. Make sure that move is highlighted and that 'number of steps' equals 1. Click **OK** and watch your piece move.

If your piece moves to the Y axis, but not the origin, change the view to SIDE or FRONT view and try again.

Now you are ready to set up the material for your bounding box.

# Set-up Job and Material Stock

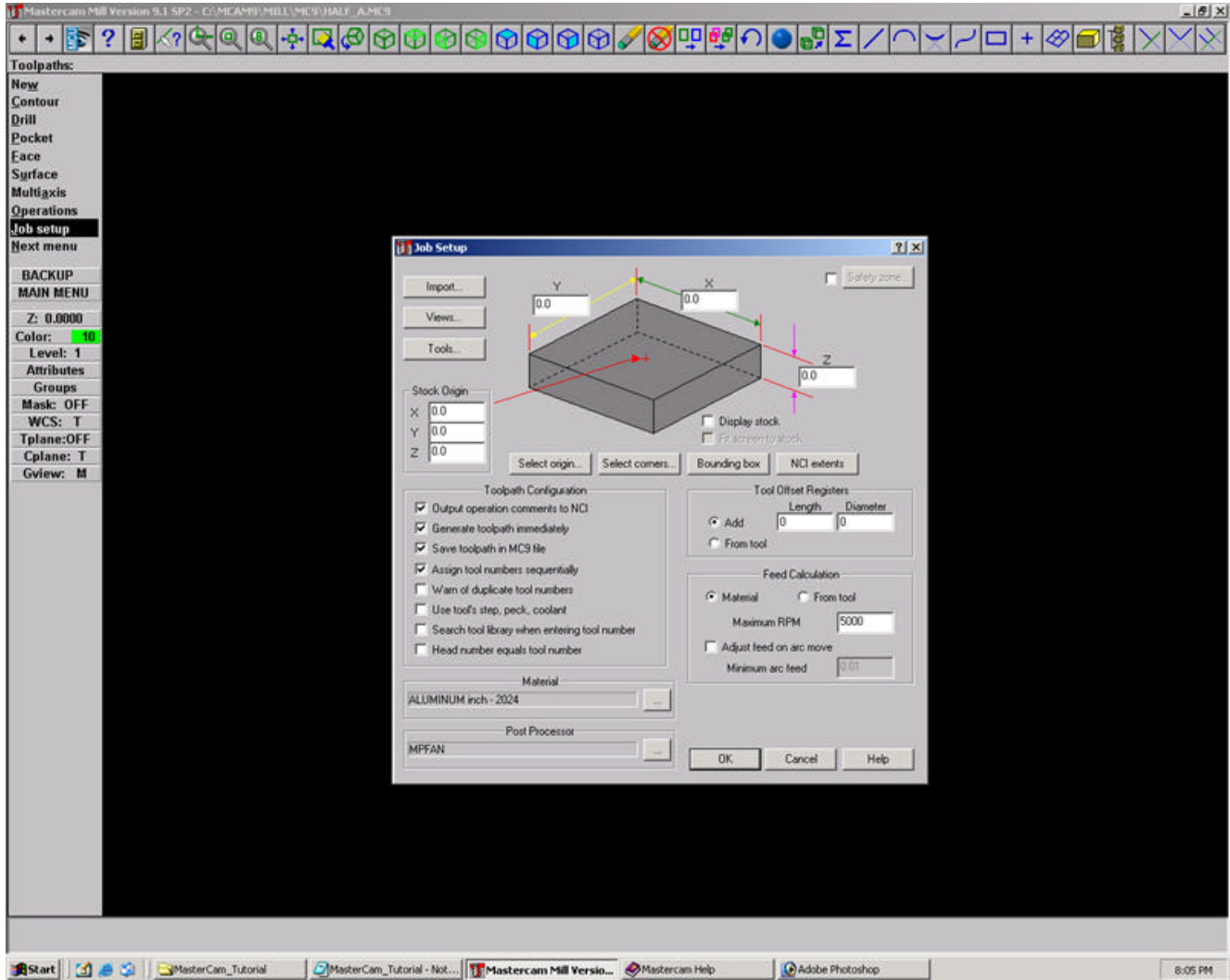
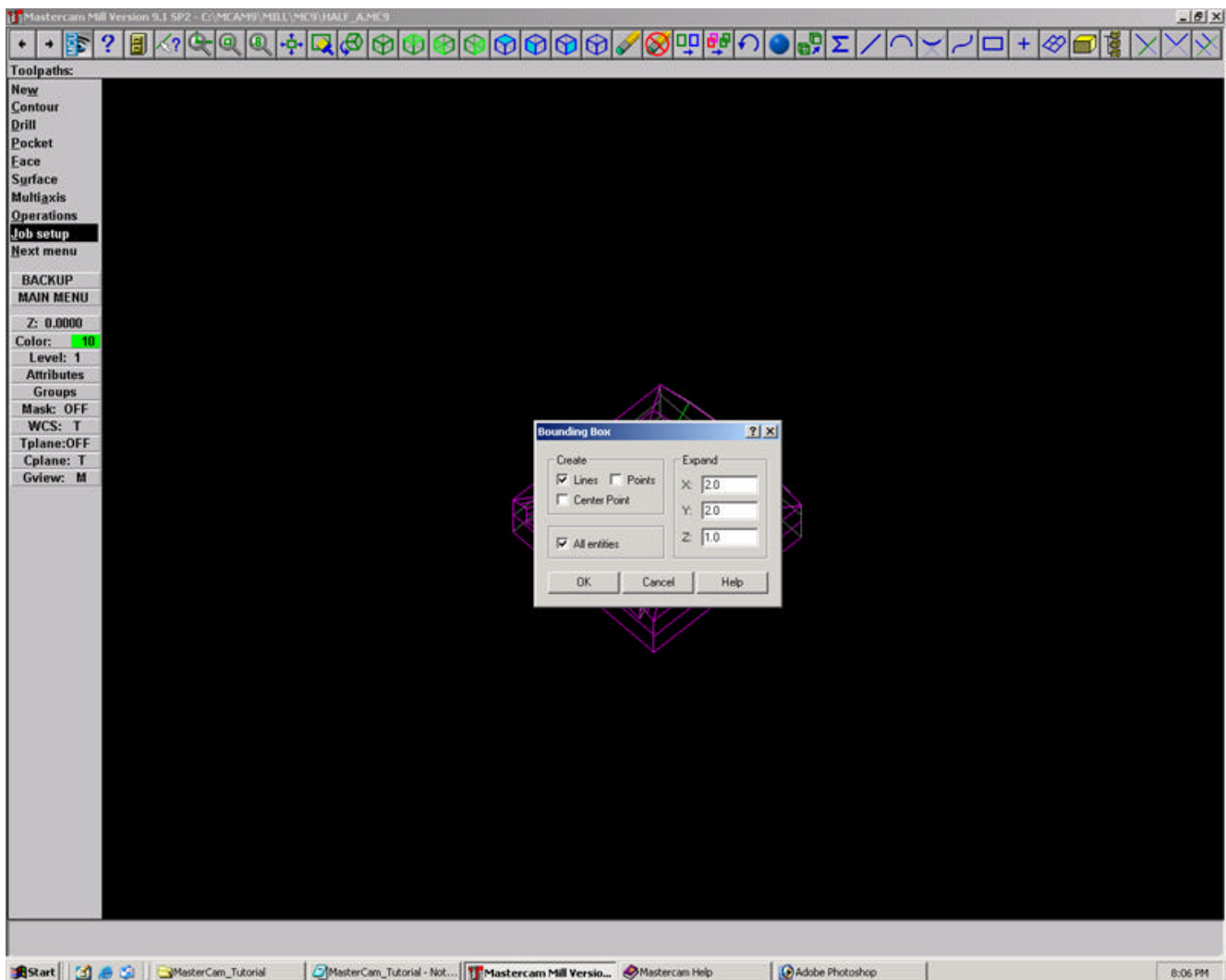


Figure 5: Job Set-Up: Setting material stock. Steps 1-2.

1. At the menu in the upper left of the screen, click "**Toolpaths**" and then click "**Job Set-up**"
2. A Job Set-up dialogue box will appear. If you have not included within your model some representation of the extents of the actual stock you will be machining, then you should set the stock here. To do, click "**Bounding Box**".



**Figure 6: Job Set-Up: Bounding box.**

3. A pop-up window will replace the dialogue box. In it, be sure you have selected "Lines" and "All entities". Nothing else. In addition, you should enter data into the "Expand" fields. This will instruct MasterCam to place a bounding box n inches offset from the edge of your model. Input data that will create a model on screen that matches how you will actually machine the stock.



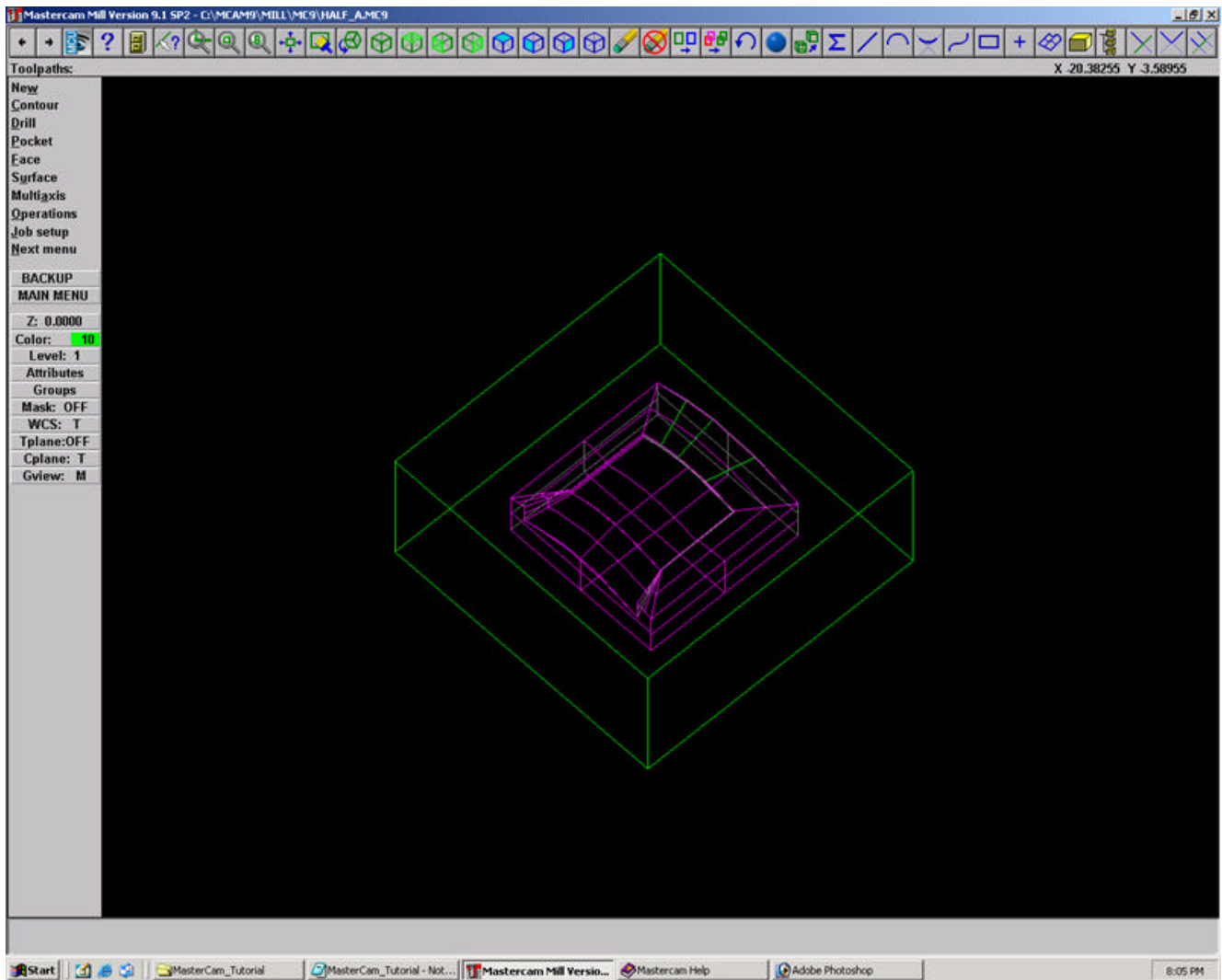


Figure 7: Job Set-Up: Bounding box. Step 5.

4. The Job Set-up dialogue box will reappear. Leave all other defaults as they are set and click **OK**.

5. You should now have your model surrounded by a bounding box that meets the specifications set.

6. To delete a bounding box, follow this click sequence:

**Main Menu/Delete/All/Lines**

Then click Main Menu again to end the sequence.

### **\*\*\*DESIGNING A ROUGH CUT**

(If you are skipping a rough cut, proceed to the next section but...

IMPORTANT: Asking a machine to perform the tool path created by a finish cut can be harmful if a rough cut has not been cut first. If you are proceeding without having first machined a rough cut, please be cautious of an extreme initial z-step. You will be able to determine this in the verify described below.)

## Select Surface(s) for Tool path

1. Click main menu at the top left to refresh the menu items and click "**tool paths**" and then "**surface**".

2. **IMPORTANT:** Ensure that the Surface settings shown at the top left are as follows (click on each variable to toggle through if you need to adjust them):

- Drive: S
- CAD file: N
- Check: N
- Contain: N

(For explanation of each of these terms, click help.)

Then click "**rough**".

3. Select the surface roughing. This is the manner in which the tool will make its cuts over the surface of the object. To learn more about the differences at this point, click on the help button.

4. If you choose parallel cut you will be prompted to tell MasterCam whether you are cutting a boss or a cavity. Do so accordingly (Boss is a positive, cavity is a negative. If you have both or a complex form, choose **unspecified**)

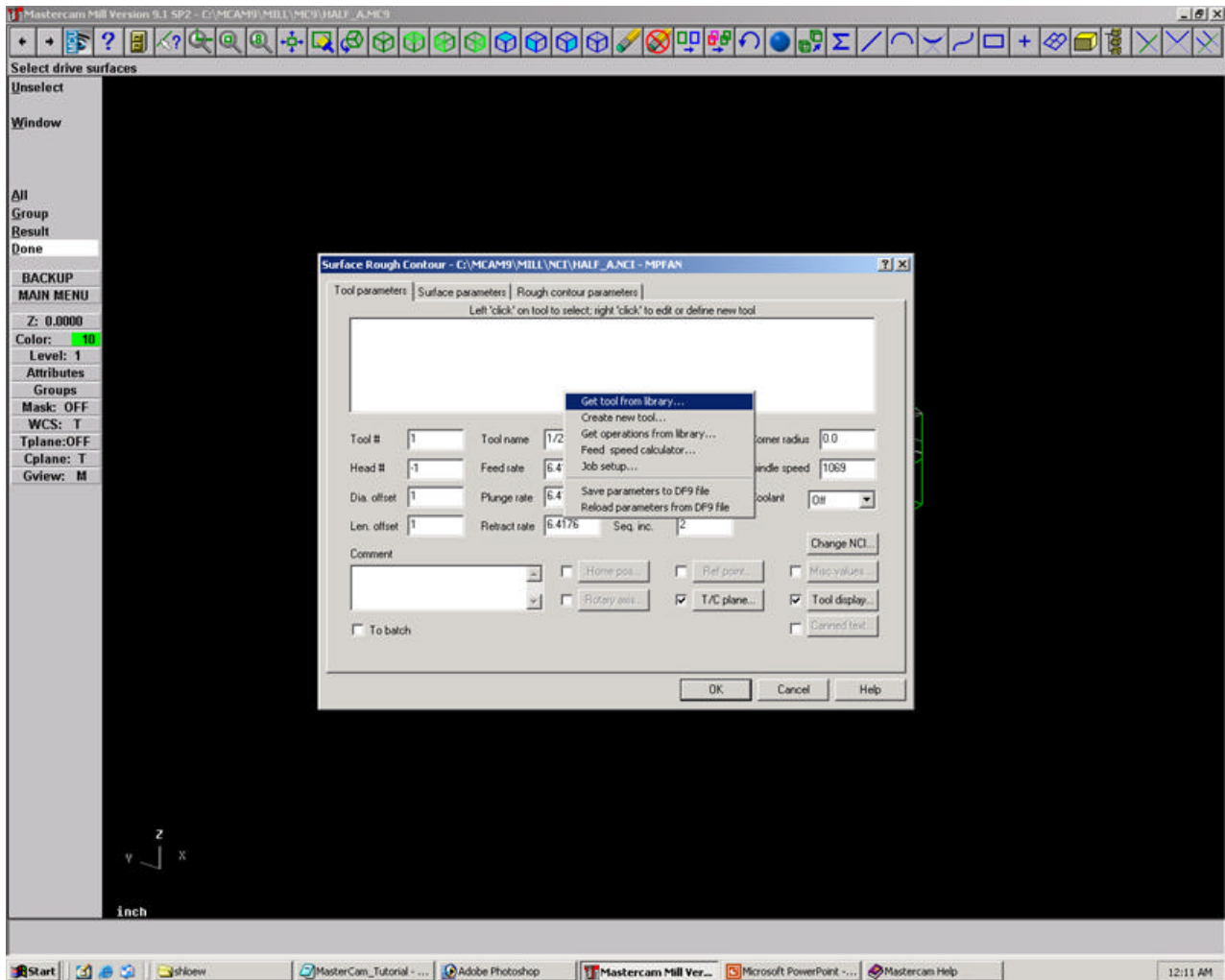
5. You will now be prompted to select the surfaces for machining. Select all surfaces feature by clicking "**All**" and then "**Surfaces**". Do not worry if this automatically selects an underside. You will verify that the machining is only cutting the desired surfaces in the next steps. Click "**Done**."

6. If you wish to select only specific surfaces rather than clicking "**All**" as described above, use the pointer and click on each desired surface. When finished selecting, click "**Done**." You may use the "**unselect**" button at the top left if you accidentally choose a surface you didn't intend to..

7. Again, once your surfaces have been selected, click "**Done**"

# Set Tool and Tool Path Parameters

1. Once done selecting your surfaces, a dialogue box will appear.
2. Under the "**Tool Parameters**" tab, right click in the open box and choose "**Get tool from Library**". Select the appropriate tool by double clicking on it. You may use the "**filter**" (button at top left of dialogue box) to help reduce the number of options shown at any one time. Once you have selected your tool, its icon should appear in the tool parameter box. You can right click on the tool now to see/confirm its specifications.



**Figure 8: Tool Parameters: Step 1-3**

3. Set following parameters (or experiment with new settings):
  - Feed Rate: 25 (for wood), 200 (for foam only, otherwise extremely dangerous)
  - Plunge Rate: 10 (for wood), 200 (for foam only, otherwise extremely dangerous)
  - Retract Rates at: 200
  - Coolant: off

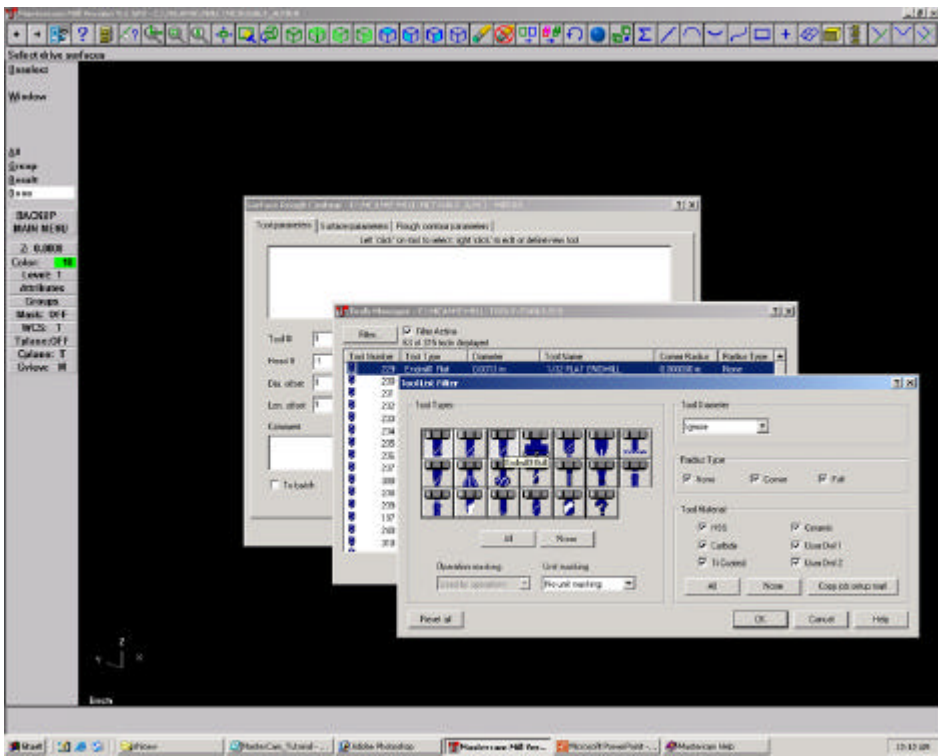


Figure 9: Tool Parameters: Step 2

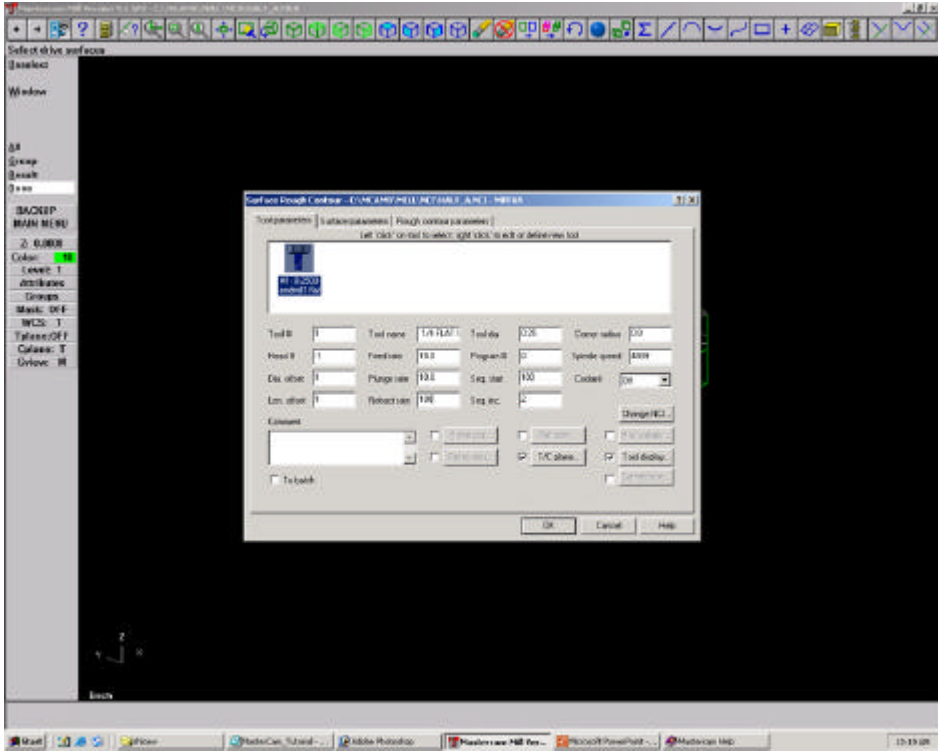
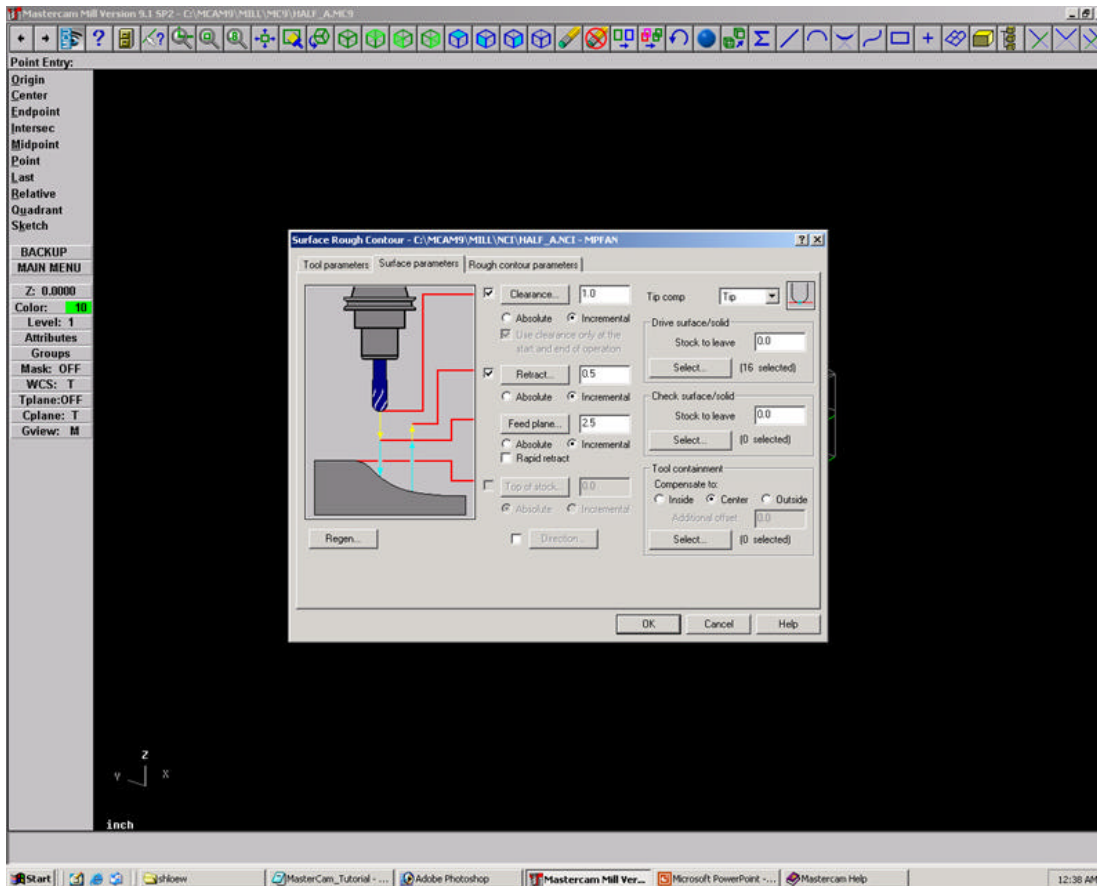


Figure 10: Tool Parameters: Step 1-3

4. Click on the Surface Parameters Tab and adjust the following parameters according to the specifics of your project:



**Figure 11: Tool Parameters: Step 4**

(NOTE: BE SURE ALL MEASUREMENTS ARE SET TO INCREMENTAL.)

- Clearance. This how high the tip of the bit will be raised off of the top of the material block when the arm travels during initial and final non-cutting movements. It will be important to forecast the height of any bracing or clamps used to hold down your material on the router bed when calculating this figure. The tip of the router must be high enough so that it won't hit any of them.

- Retract. This is the height the tip of the bit will be raised off of the top of the material block in between cutting movements.

- Feed Plane. This is distance off of the machined material to which the feed rate will continue and after which the Plunge Rate will begin. In other words, the tool will move very quickly when it is above this height, and will move at the feed rate when it is below this height.

- Tip comp. This is the point on the bit from which the cutting measurements are drawn. You do not want to use center comp.

AGAIN, BE SURE ALL MEASUREMENTS ARE SET TO INCREMENTAL.

5. Click on the Rough Contour Parameters tab and adjust the settings here. You should consider the following but are encouraged to experiment with all settings to obtain a desired tool path:
- "**Maximum Stepdown**" size. This will be determined by the material you are machining, the tool bit being used and other factors. Experiment with this and other settings in this tab and the tab if you'd like. Generally, you should not try to cut down any further than the width of your cutting tool. A 1/2 inch diameter cutting tool can cut 1/2 inch deep on every roughing pass, for example. Cutting more than this will cause tool bits to break off under general use.
  - "**Cutting Method**". It is recommended that you use zigzag unless you are using a particular material with a grain that prefers a certain bit approach.
  - "**Max Steptover**" - this is how far the tool will move over when it changes direction at the end of each line. A large distance will cause large scallops if you are using a spherical bit. A small distance will take much longer to cut as there will be many more cutting passes, but the shape will be smooth. For a roughing pass you can usually use a large number, up to half the width of your cutting tool.
  - Once finished, click **OK**.
6. If a pop-up window notifies you of sharp internal corners, you can decide if you wish to proceed or not. This is indicating that your surface may not be machinable given the tool parameters and model geometry. If you wish to remodel, then click **no** and close out and remodel in the original modeling software. If you do not care about this then click **Yes** and proceed. See the section above on cleaning-up your model for more details on testing for this scenario. Again, click **YES** to proceed.

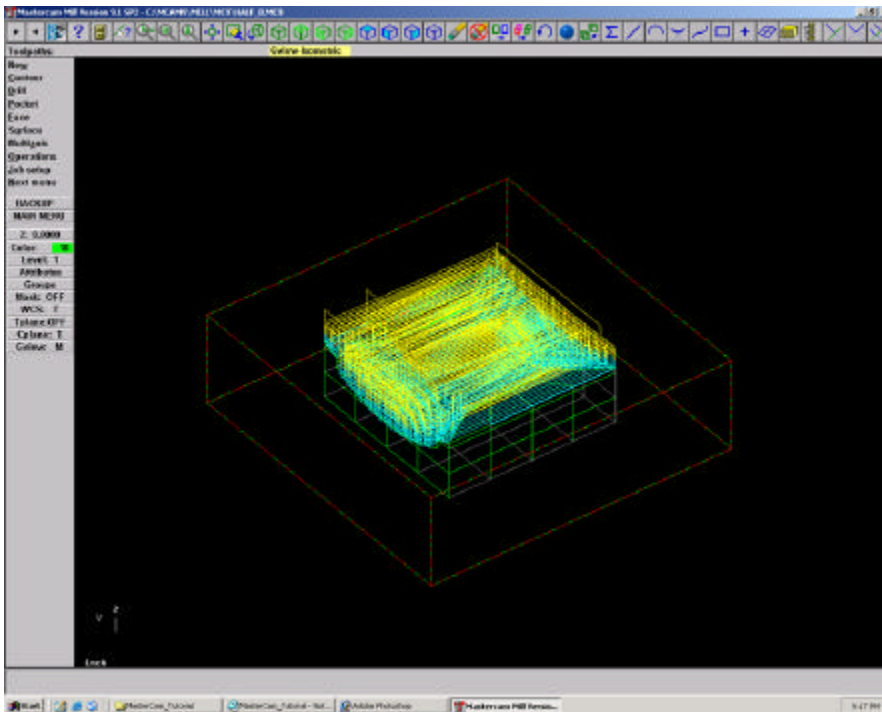


Figure 12: Tool path drawn. Step 7.

7. You will then observe MasterCam automatically defining a tool path for the given parameters you set-up. Gray are cut lines, Blue are plunges, yellow are traverses, and red are intersections/overlaps. Once it completes this cycle, click as follows:

**Main Menu/Tool Path/Operations**

This will open the Operations Manager where you will verify the tool path.

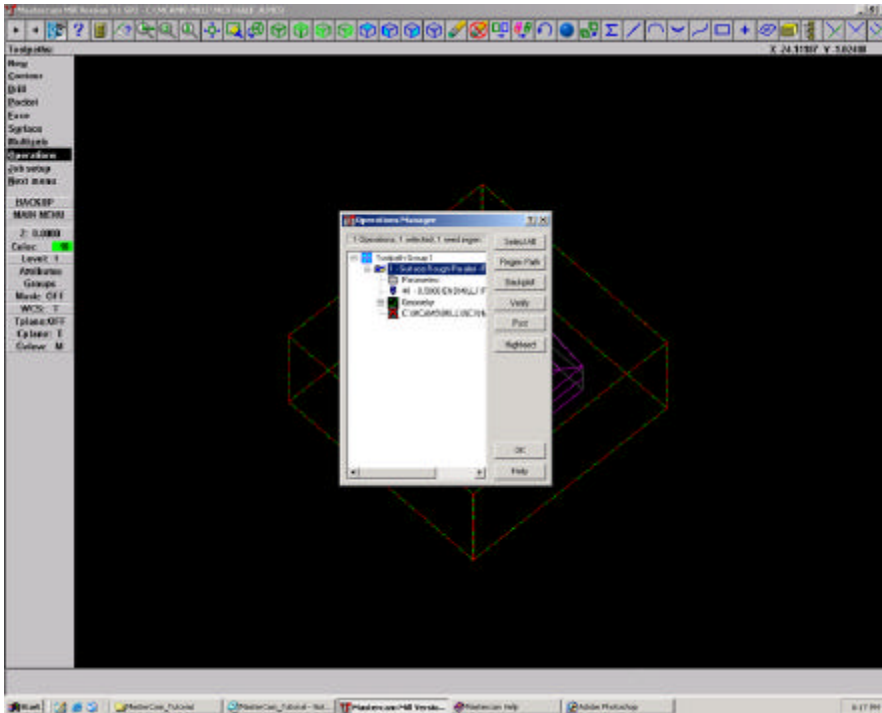


Figure 13: Operations Manager. Step 7.

## Verify & Post NC file

1. With the operations Manager dialogue box open (see last step in section above) click Verify at the right. This will launch the in-program verify which will mimic the tool-path for the operation you have just established. You will use the verify as an initial proof of the settings. Observe the following:

- Is the bit machining the material block or is it off in space?
- Is the z-step set to a proper depth?
- Is the incremental step (x and/or y) an appropriate amount for the affect I'm after?
- Is the surface correctly being machined?
- Is the tool machining too far over (laterally - x and/or y) from the material stock?

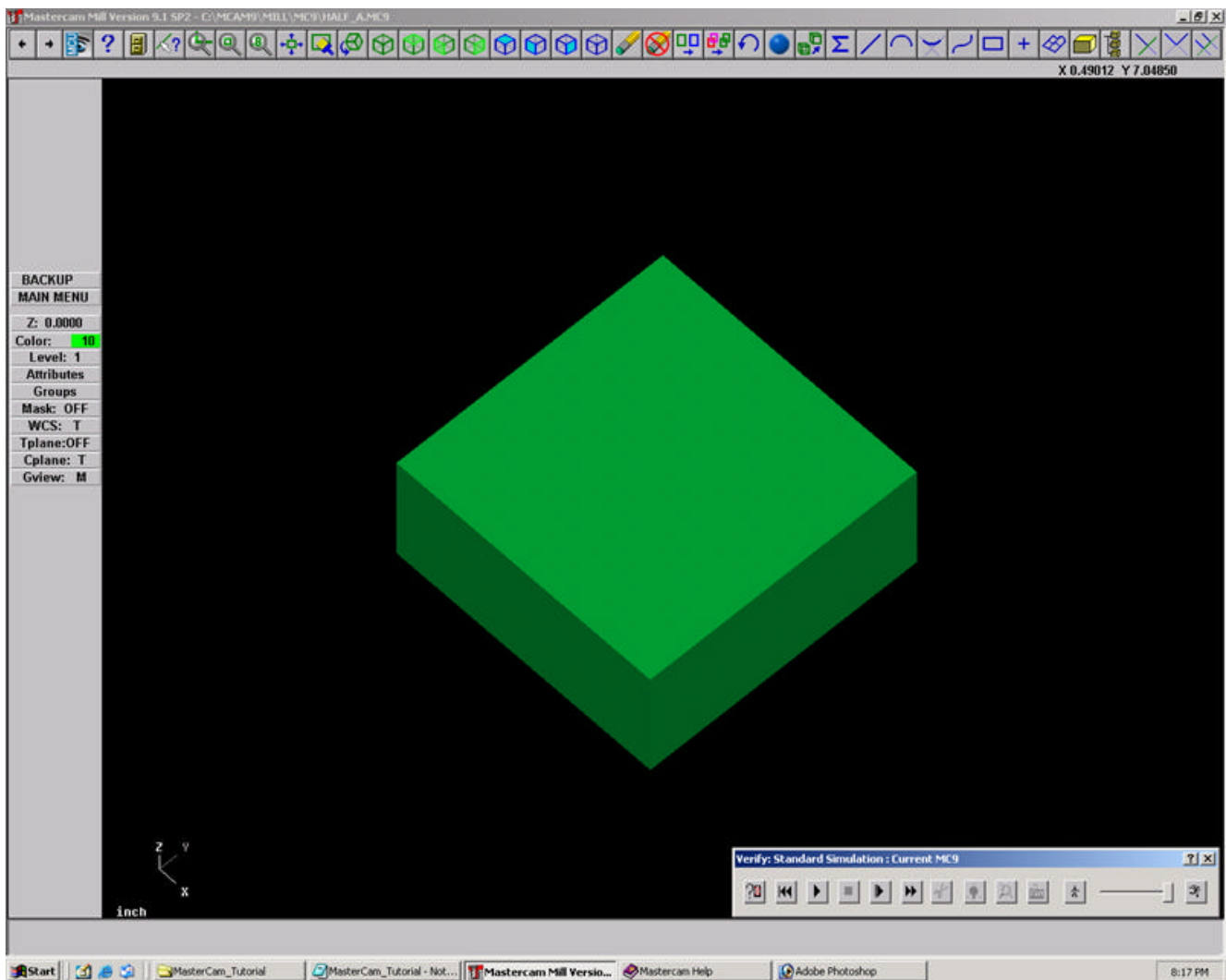


Figure 14: In-program Verify tool. Step 1-2.

2. To run the verify, click the **play** button.

3. If you are unsatisfied for any reason, return to the Operations Manager and click on the "**Parameters**" line to return to the tool parameters dialogue box. Make any necessary adjustments and say **OK**.

IMPORTANT: If you have altered the parameters, you must return to the **Operations Manager** and click "**Regen**". This will regenerate the tool path. Then repeat the steps outlined above to re-verify the new tool path. Repeat these steps until you are satisfied with the tool path demonstrated in the verify.



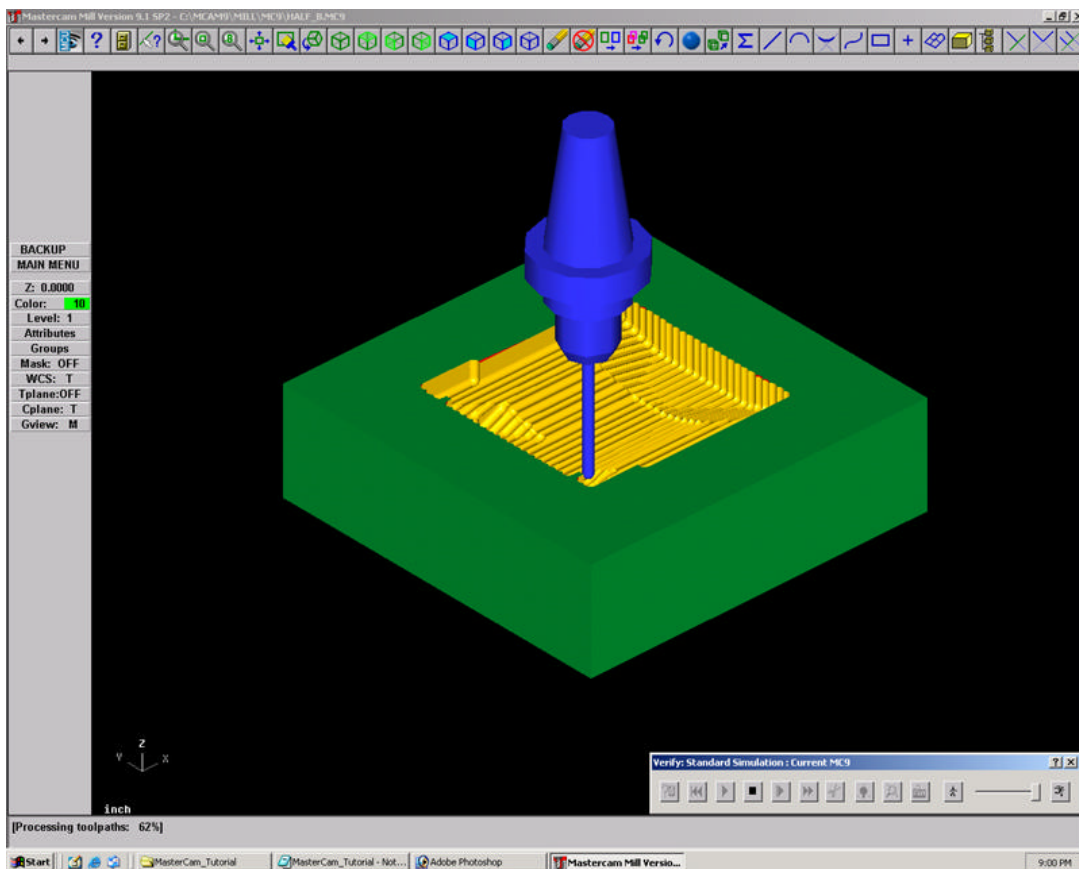


Figure 15: In-program Verify tool. Step 1-4.

4. If you are satisfied with the verify, close the verify bar by clicking on the "x" at the top right. This will reopen the **Operations Manager**. From within the **Operations Manager**, highlight the tool path in the window on the left and click on the "Post" button on the right.

5. This will open the **Post Processing** dialogue box. You should use the **XYZ** postprocessor. If it is not shown in the dialogue box, choose "**change post**" and choose it from the list (it will be the first one on the list). Click on **save NCI**, as well as **save NC**. You do not need to click the edit box. When prompted, save the file to your desired location. **DO NOT SAVE FILES ON THE CAD/CAM Computers**. They will be deleted regularly.. Be sure your file is labeled as an NC file type.

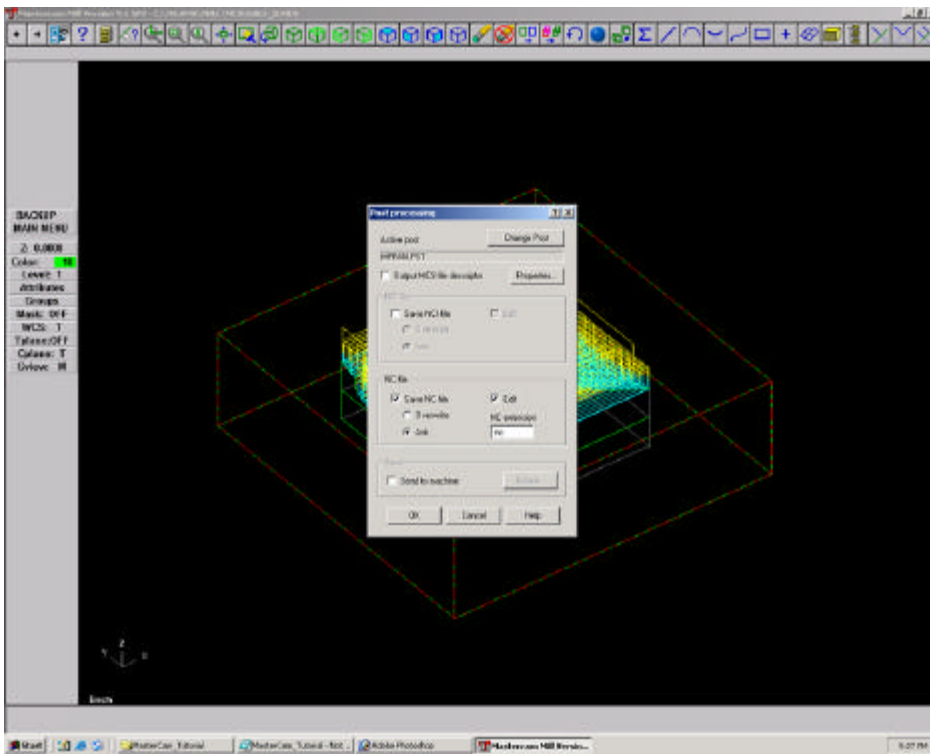


Figure 16: Post-Processing dialogue box. Step 5.

6. Saving the file will cause your G-Code to be written. Once it is written you may open the file using any text editor, but this may not be necessary.

7. You now have a rough cut tool path ready for uploading to the mill/router.

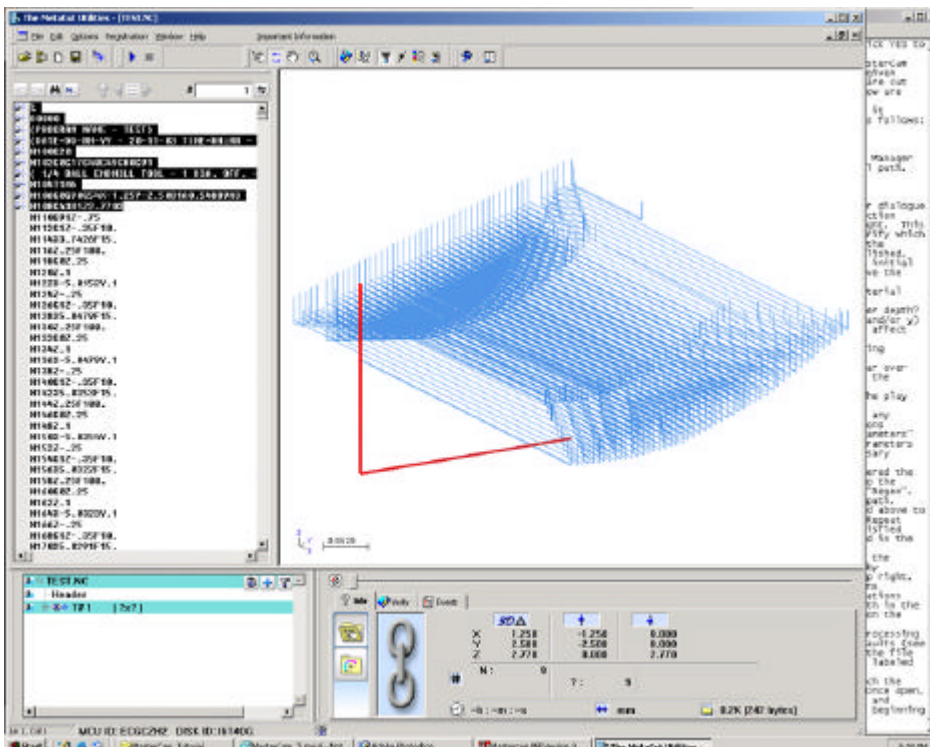


Figure 17: MetaCut Utilities. Step 6-7.

## FINISH CUT

(IMPORTANT: Asking a machine to perform the tool path created in a fine cut can be harmful if a rough cut has not been cut first. If you are proceeding without having first machined a rough cut, please be cautious of an extreme initial z-step. You will be able to determine this in the verify described below.)

Select Surface(s) for Tool path

1. Click main menu at the top left to refresh the menu items, then click "tool paths" and then "surface"

2. **IMPORTANT:** Ensure that the Surface settings shown at the top left are as follows (click on each variable to toggle through if you need to adjust them):

- Drive: S
- CAD file: N
- Check: N
- Contain: N

Then click "Finish"

3. Select the Surface Finishing. This is the manner in which the tool will make its cuts over the surface of the object. To learn more about the differences at this point, click on the help button.

4. You will now be prompted to select the surfaces for machining. Select all surfaces feature by clicking "All" and then "Surfaces". Do not worry if this automatically selects an underside. You will verify that the machining is only cutting the desired surfaces in the next steps. Click "Done."

5. If you wish to select only specific surfaces, rather than clicking "All" as described above use the pointer and click on each desired surface. When finished selecting, click "Done." You may use the "unselect" button at the top left.

6. Again, once your surfaces have been selected, click "Done"

## **Set Tool path and Parameters**

1. Follow the instructions described above for setting the tool path and parameters for a rough cut. Note that the finish cut offers fewer parameter options as the finish cut is dealing principally with only the surface of the model (rather than the whole unmachined stock/block). In addition, different Surface Finishing have different parameters associated with them.

## **Verify & Post**

1. IMPORTANT: To verify a fine cut when machining it following a rough cut, it is important that you run both rough and fine cut consecutively in the verify to get a true picture of the complete tool path. To do this, open the Operations Manager and select BOTH rough cut and fine cut (hold down the control key to select more than one). Then click "Verify" at the right and run the verify by clicking on the play button.

2. To modify the tool parameters, follow the instructions described in the rough cut sequence.

IMPORTANT: if you are adjusting parameters and re-verifying, you must first regenerate the tool path by clicking on the "regen" button from within the Operations Manager and then re-run the verify.

3. To post the fine cut, follow the same instructions as described above in the rough cut.

IMPORTANT: Be sure that only the fine cut in the Operations Manager is selected when you click on the Post button. If both rough cut and fine cut are selected, you will create g-code for both consecutively. It is recommended that you post each separately so as to have more control at the machine.

4. Once you have written the G-Code you are ready to take it to the Toolpath program that acts as an interface to our AXYZ router, or to our milling machine software.