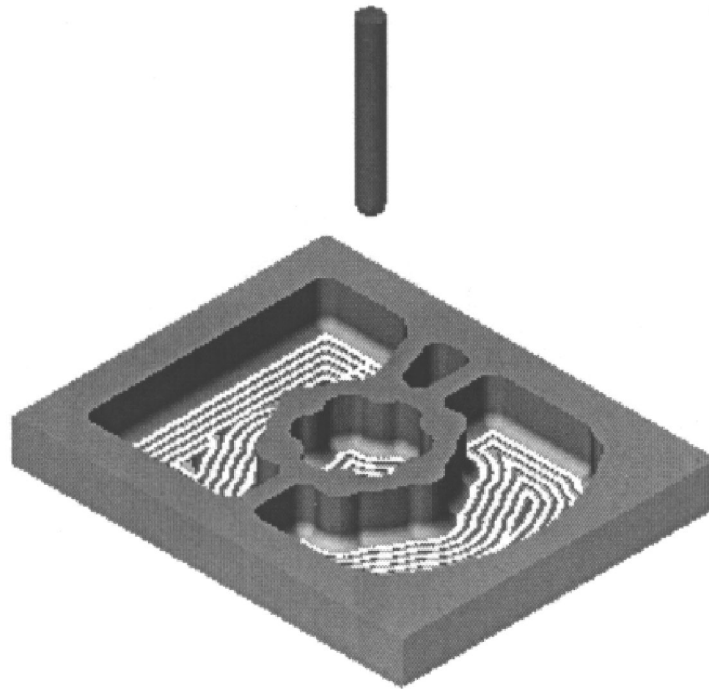


# SURFCAM<sup>®</sup>

# Tutorial



*An Introduction to SURFCAM:*  
A 2D Numerical Control Project




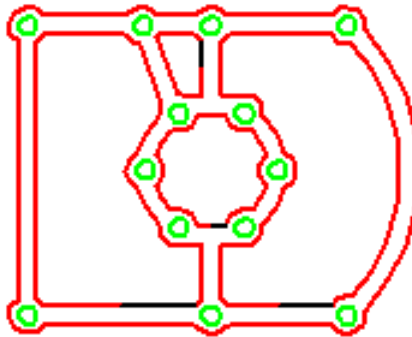
Tech Ed Concepts, Inc. • 550 Pembroke Street • Pembroke, NH 03275  
Tel. (800) 338-2238 • Fax (603) 224-8324  
Web: [www.TECedu.com](http://www.TECedu.com) • E-mail: [Sales@TECedu.com](mailto:Sales@TECedu.com)

**PART 4: A COMPLETE PROJECT****3.13 A 2D NC PROJECT**

1. Open a file and create a new Setup Section.
2. Create a pocket.
3. Create the toolpath.
4. Verify the toolpath.
5. Create the NC code.
6. Transfer the NC code to the machine.

**3.13.1 Open a file and create a new setup section**


1.  Open the original 2dribs.dsn file.

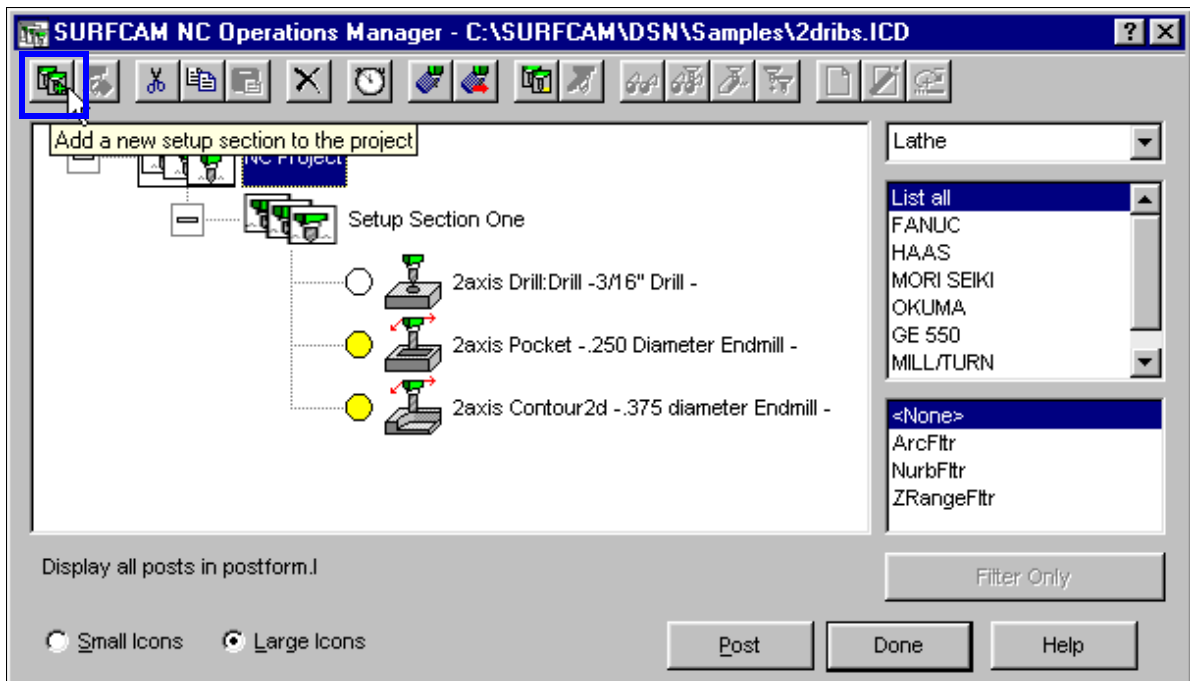


Tip If you saved the dimensions you created in [Create the dimensions on a design, page 79](#), press the delete key then click on each dimension to delete the dimensions.

- Press CTRL+7 to change to Isometric view.



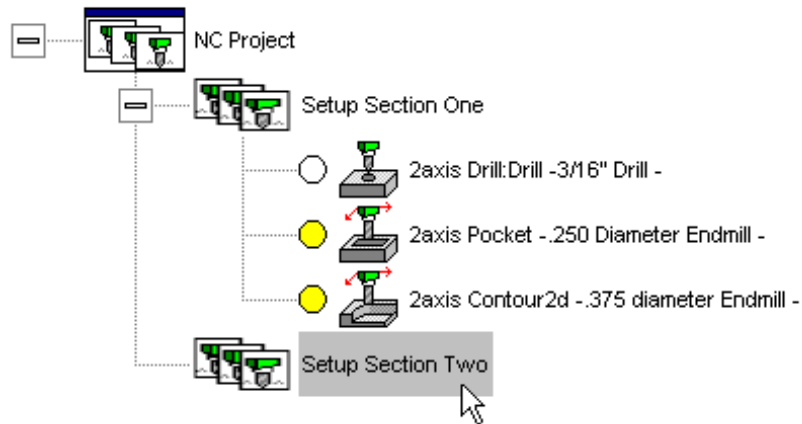
- Click the *NC Operations Manager* button on the SURFCAM toolbar to display the dialog box. 
- Click the *Add a new setup section to the project* button on the toolbar of the NC Operations Manager dialog box.



- Enter the name Setup Section Two.



- Highlight Setup Section Two in the NC Operations Tree and click the *Done* button.



### 3.13.2 Select the machining mode and identify the geometry to cut

- Click the *NC > 2 Axis > Pocket* command.
- SURFCAM displays the *NC > 2 Axis > Select Chain* menu and prompts you to select the element to begin.

**Note** Use the chain process to define the boundary on the part for the removal of material. The boundary defines the pocket.

There are a number of different methods to chain the geometry to cut the pocket. Use the *Auto* command for the fastest method.

Click the *Select Chain > Auto* command.



- Click the **Select > Visible** command.



- After the geometry is highlighted, click the **Select > Done** command.

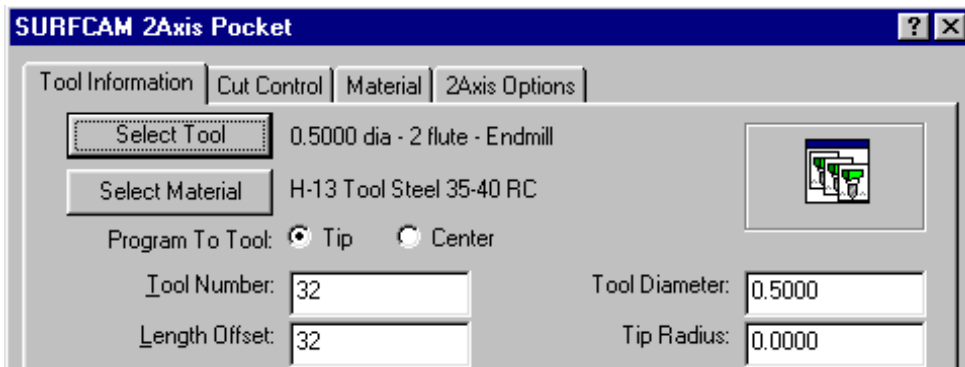


- Click the **Select Chain > Done** command.

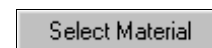


### 3.13.3 Create the toolpath

After the geometry is identified, SURFCAM displays the **2 Axis Pocket** dialog box.



- Click the **Select Tool** Button.
- Click the third button on the toolbar to select an Endmill tool.
- Select the .5 diameter 2 Flute Endmill tool.
- Click the **OK** button on the **Select Mill Tool** dialog box.
- Click the **Select Material** button.
- Select a material.
- Click the **OK** button on the **Select Material** dialog box.



8. Now click the *Cut Control* tab to define a depth to cut the pocket toolpath.

The screenshot shows the SURFCAM 2Axis Pocket dialog box with the 'Cut Control' tab selected. The 'On Sides' and 'In Z' columns are highlighted with a blue box, indicating the values for 'Pocket Depth' and 'Increment'.

Parameter	On Sides:	In Z:
Amount To Remove:		0.500000 <i>Pocket Depth</i>
Rough Spacing:	0.200000	0.250000 <i>Increment</i>
Finish Passes:	1	0
Finish Spacing:	0.020000	0.020000
Stock To Leave:	0.000000	0.000000

Other parameters shown in the dialog include:

- Cutting Method: Climb
- Pocket Cut Mode: Spiral
- Curve Tolerance: 0.001000
- Rapid Plane: 1.000000
- Plunge Clearance: 0.220000
- Geometry: Top
- Taper Angle: 0.000000
- Cutter Compensation: Offset
- Leadin Move: None
- Leadout Move: None
- Plunge Type: Plunge
- Machine Left-over Material:
- Enable High Speed Machining:
- Minimum Radius: 0.002000
- Maximum Radius: 1.000000
- Disable for finish passes:

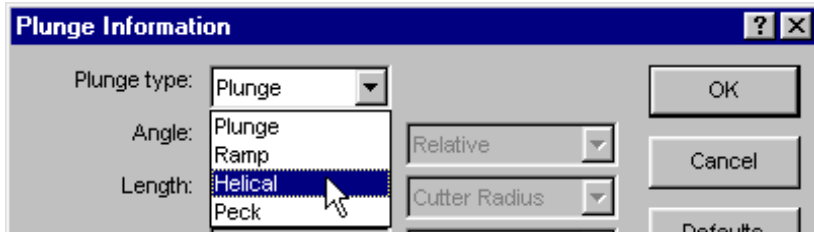
The values in the *On Sides* column refer to movements in the X and Y directions.

The values in the *In Z* column refer to movements in the Z direction.

9. Enter the Pocket Depth of 0.5 and enter the Increment Value of 0.25 for equal passes as shown in the previous figure.
10. Click the *Plunge* button.

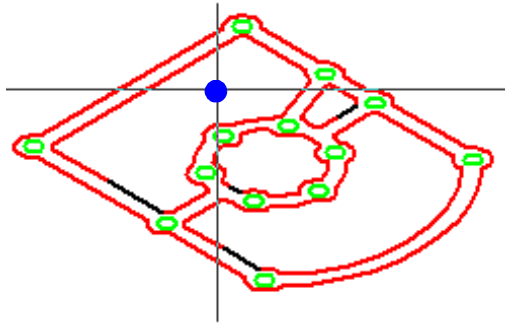


11. Select the *Helical* option for the *Plunge Type*.

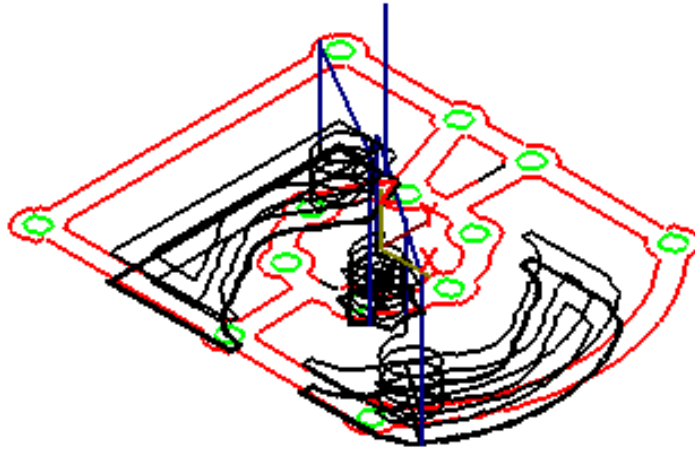


12. Click the *OK* button on the *2 Axis Pocket* dialog box.
13. A prompt tells you to click the inside of a pocket. Click the inside of any one of the pockets.

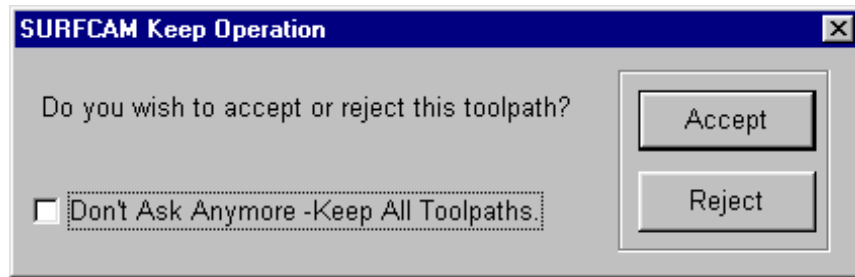
**Tip** SURFCAM draws the toolpath in all pockets that the .5 diameter 2 Flute Endmill tool can fit in because you clicked the *Select Chain > Auto > Visible* command.



14. SURFCAM creates the toolpath. You can see that the tool was not the correct size to cut the smallest pocket.



15. When the *Keep Operation* dialog box is displayed, click the *Accept* button.



### Demo Version Reminder

You cannot save a toolpath in the Demo Version of SURFCAM.

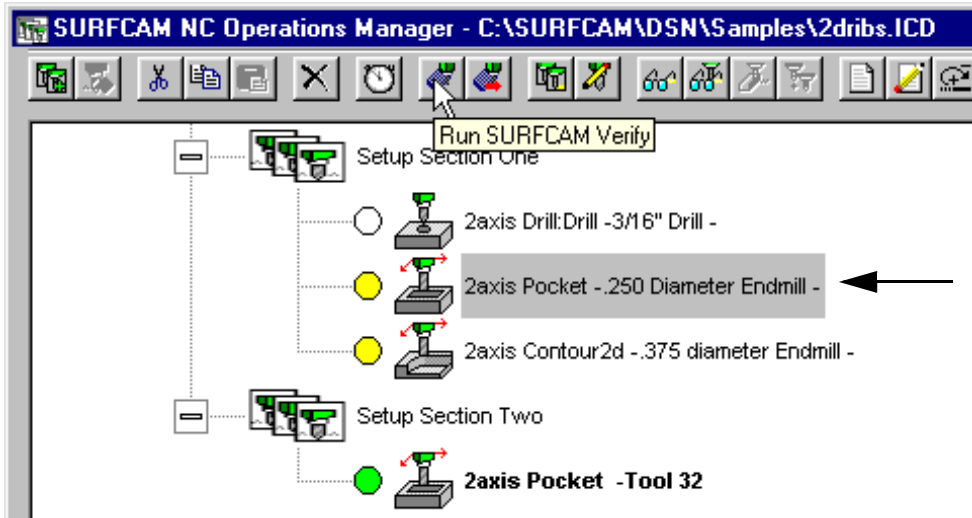


### 3.13.4 Verify the toolpath

1. Click the *NC Operations Manager* button on the SURFCAM toolbar to display the dialog box.
2. You cannot save a toolpath in the Demo version.



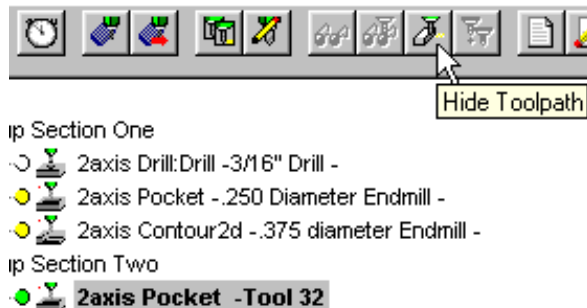
Highlight the Pocket operation Setup Section One so that the example is correct for any version of SURFCAM that you use.




**Note** The toolpath that you created is displayed in bold print because that toolpath is visible on your screen.

#### Optional

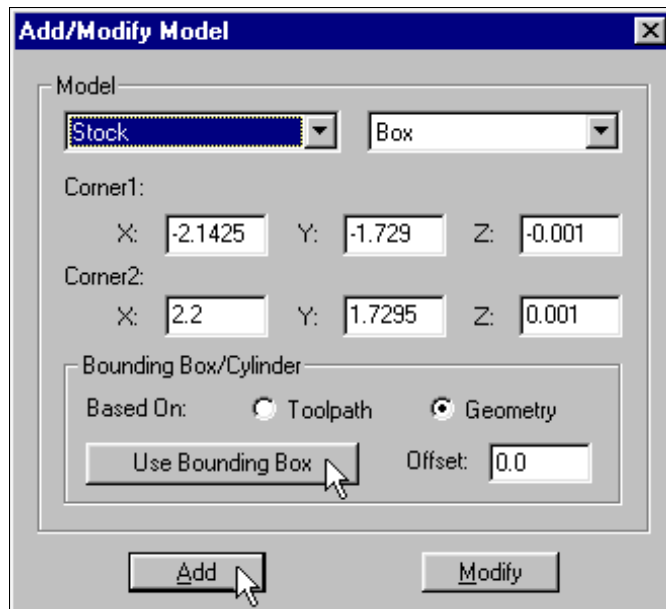
- Highlight the toolpath that you created in Setup Section Two.
- Click the *Hide Toolpath* button. The toolpath is no longer visible.
- Highlight again the Pocket operation in Setup Section One.



3. Click the *Run SURFCAM Verify* button. 
4. The Integrated Verify menu is displayed. Click the *Add Model* button.



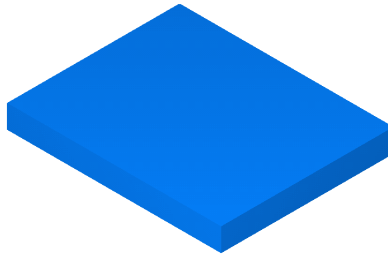
5. The *Add/Modify Model* dialog box is displayed. Click the *Use Bounding Box* button.



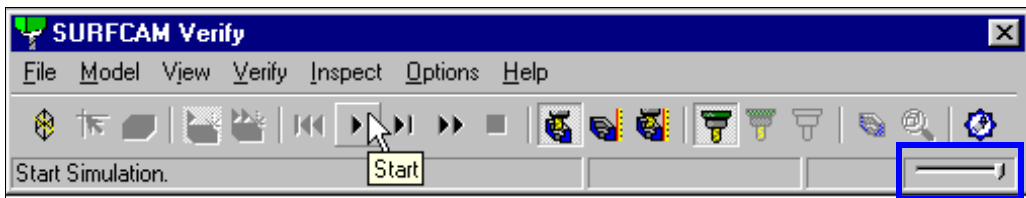
6. Click the *Add* button.
7. Click the *Verify* button.



- The part is now contained in a solid box of material.



- Click the *Start* button.

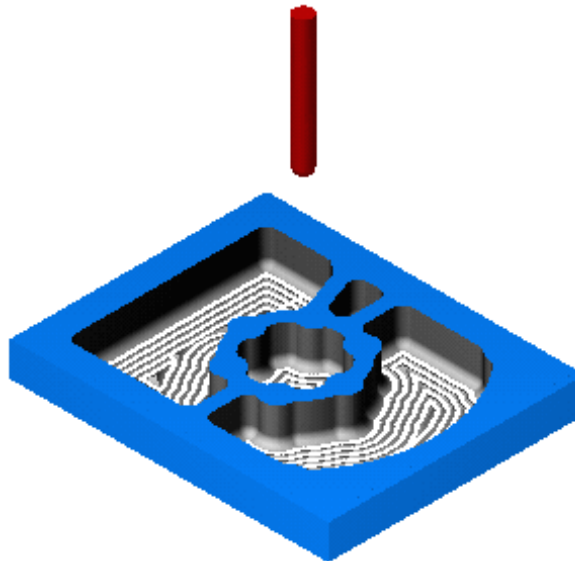


- SURFCAM simulates the cut of the part. See the *Slider Bar* in the graphic.




Move the tab to the left or to the right any time during the verification.

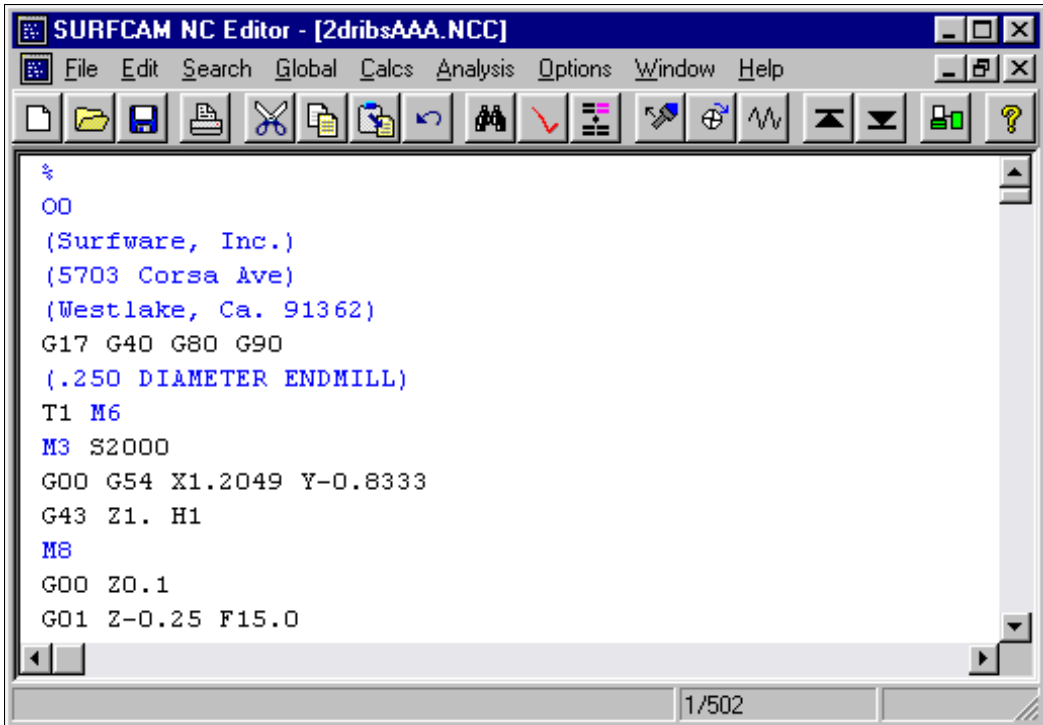
Move the tab to the right for a faster verification. Move the tab to the left for a slower verification so that you can get a better view of a section.



- Exit the Verify program when the verification is complete.

### 3.13.5 Create the NC code

1. Open the *NC Operation Manager* dialog box.
2. Make sure that the 2 Axis Pocket toolpath is  highlighted in the NC Operations Tree.
3. Select a machine from the list on the right of the dialog box.
4. Click the *Post* button.
5. SURFCAM displays the NC code in the *NC Editor* program.



The screenshot shows the SURFCAM NC Editor window with the following text:

```

%
O0
(Surftware, Inc.)
(5703 Corsa Ave)
(Westlake, Ca. 91362)
G17 G40 G80 G90
(.250 DIAMETER ENDMILL)
T1 M6
M3 S2000
G00 G54 X1.2049 Y-0.8333
G43 Z1. H1
M8
G00 Z0.1
G01 Z-0.25 F15.0

```

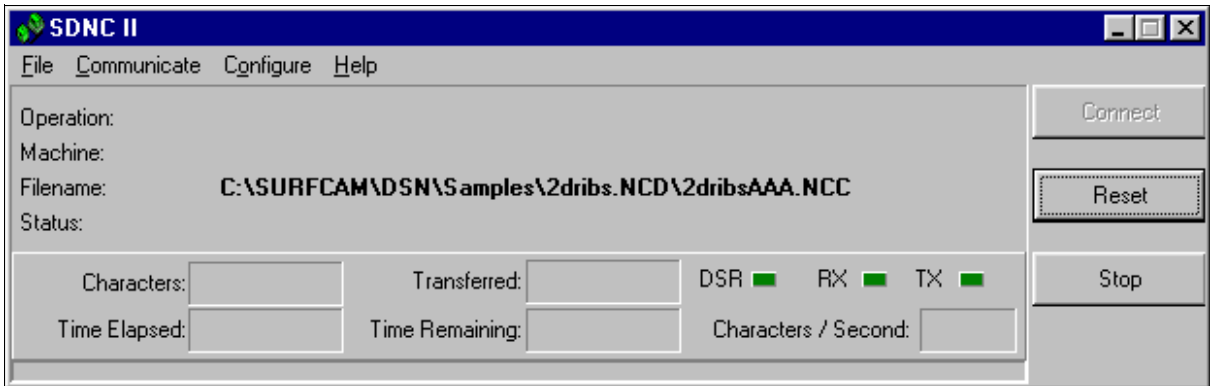
The status bar at the bottom right of the window displays "1/502".

After you edit this file, you can transfer the code to the machine. Keep the editor window open for the next section.

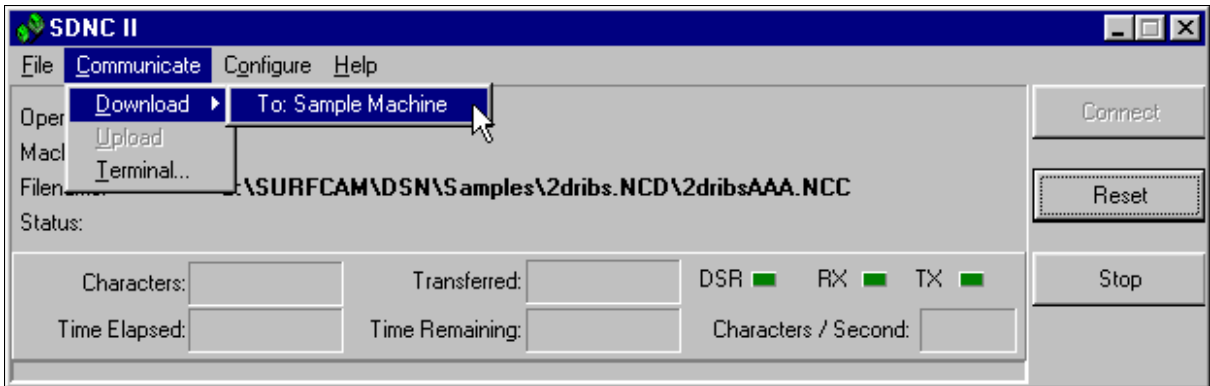
### 3.13.6 Transfer the NC code

You can use the SURFCAM DNC (SDNC) program to transfer the NC code to the machine.

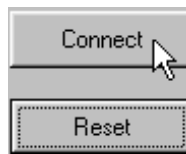
1. Click the **SDNC** button on the NC Editor toolbar. The **SDNC II** dialog box is displayed.



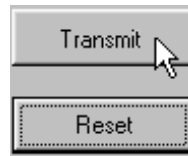
2. Click the **Communicate > Download > To: Sample Machine** command.



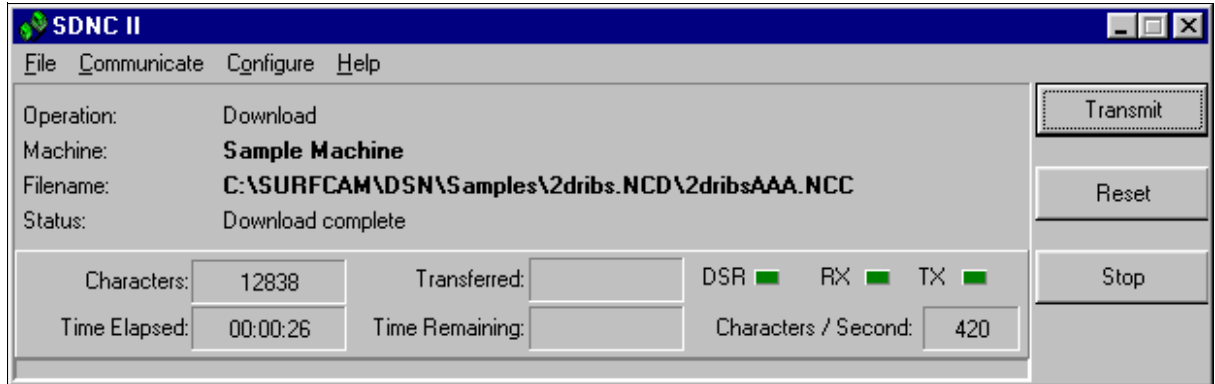
3. Click the **Connect** button.



- Click the *Transmit* button.



- Note the number of *Characters* that were transmitted and the *Time Elapsed*.



- See [the online SURFCAM DNC chapter](#) for complete information about SDNC.