Pro/MFG Volume Milling Tutorial

OBJECTIVES:

The Manufacturing module in Pro/E is used to generate machine code to produce parts. This tutorial will focus on Pro/MFG Volume Milling. A volume milling NC sequence removes the material inside a Milling Volume slice by slice. All slices are parallel to the retract plane. Some typical applications for volume milling are:

- Facing down the workpiece

- General material removal on the side of the workpiece

- Rough milling of a vertical slot or cut, or a blind slit with islands

- Pocketing finishing by using the ROUGH_OPTION parameter value PROF_ONLY

PROCEDURES:

Index:

Start:

A. Produce the part model. The part consists of two features, together forming a block (4" x 8" x 1.5") with raised letters "CAD" as shown in Figure 1.12.

1. Start the Pro/E program. Windows menu items, Start → All Programs → PTC → Pro ENGINEER→ Click on the icon [icon] Pro ENGINEER.

2. Set working directory. File → Set Working Directory, select working directory.

3. Create the part name. Pro/E main menu, File → New, select Part in the New window, enter part name: CAD.

4. Start of the part. The part contains some features already. The main graphics area shows 3 datum planes and a coordinate system as shown in Figure 1.1.
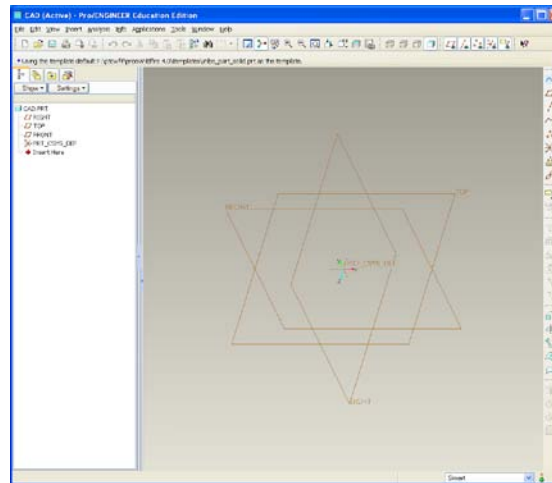
Figure 1.1 Start of the Part

5. Create a 4" x 8" x 1.5" rectangular block as shown in Figure 1.8.
   - Choose INSERT → EXTRUDE from the menu. You should see a new toolbar called dashboard appear as shown in Figure 1.2.
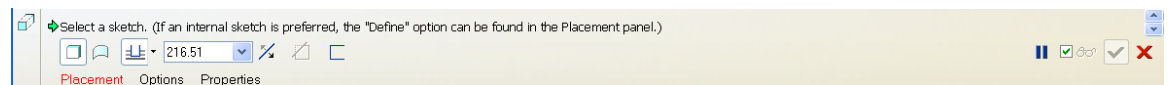

Figure 1.2 The Dashboard

   - Click on the placement on the extrude dashboard (Figure 1.3) and select define. The sketch dialog window appears as shown in Figure 1.4.
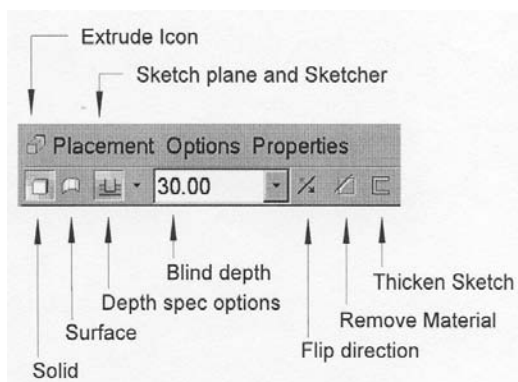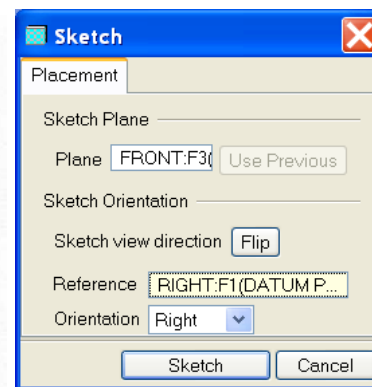

Figure 1.3 The Extrude Dashboard   Figure 1.4 Sketch dialog

   Choose the datum place FRONT by clicking on it in the graphics window. Accept the default on the sketch dialog window and just click on the Sketch button. References sub-window pop up as shown in Figure 1.5 and click close to start sketch.
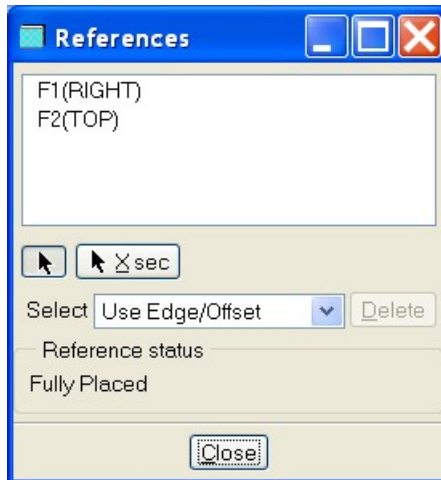
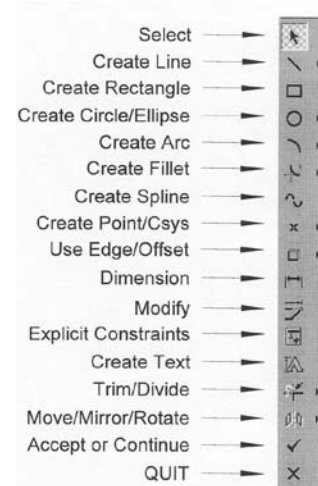Figure 1.5 References dialog   Figure 1.6 The Sketcher toolbar

- Choose Create Rectangle icon on Sketcher Toolbar (Figure 1.6) to sketch a rectangle for extrusion in plane FRONT by clicking on the bottom-right and top-left corners of the rectangle in the drawing windows. Click middle mouse to finish drawing the rectangle.
- Modify the dimensions to 4 x 8 by double clicking on the dimensions on the rectangle. To end sketching choose Accept icon on Sketcher Toolbar (Figure 1.6) and click OK in the Section dialog.
- Extrude the rectangle to form the solid block. Enter depth value as 1.5 into the depth field of the extrude dashboard (Figure 1.3) and click the Accept tick to finish (Figure 1.7).
- View the rectangular block as shown in Figure 1.8. View → Orientation → Standard Orientation.
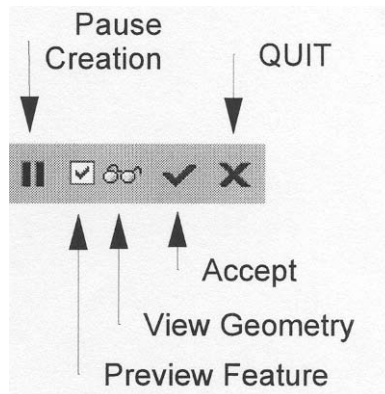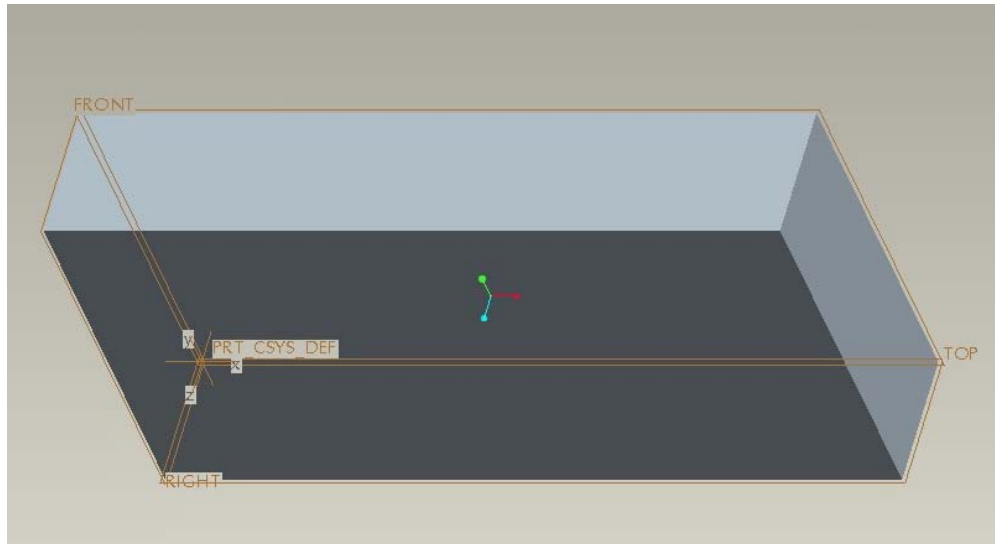

Figure 1.7 Dashboard controls

Figure 1.8 Block

6. Create 2" x 6" x 0.25" letters "CAD" on top of the block as shown in Figure 1.12.
   - INSERT → EXTRUDE from the menu. The dashboard appears as shown in Figure 1.2.
   - Click on the placement on the extrude dashboard (Figure 1.2) and select define. The sketch dialog window appears as shown in Figure 1.9. Choose top surface of the block as sketch plane and accept the default on the sketch dialog window and click on the Sketch button.
   - Produce letters. Sketch → Text from the menu and pick two points on the sketch plan to determine the height of the text and enter letters CAM as shown in the figure 1.10. Modify the size of the text to 2 x 6 and locate the text to the center of the block as shown in figure 1.11.
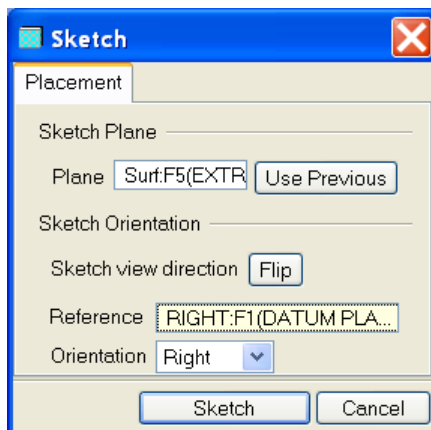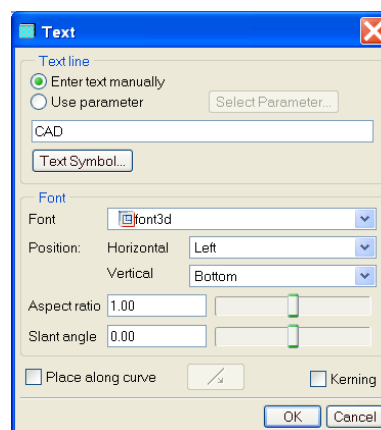

Figure 1.9 Sketch dialog
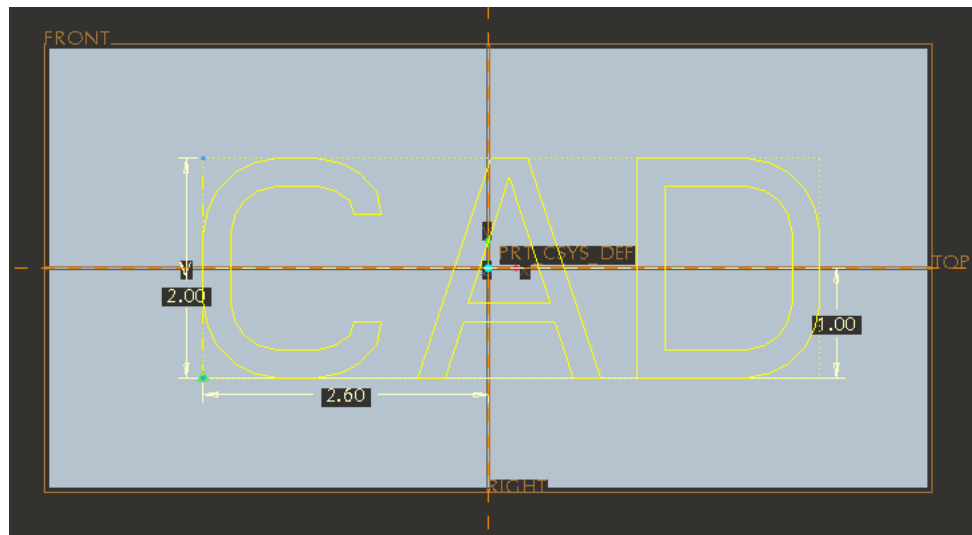

Figure 1.10 The Text dialog

Figure 1.11 Letter CAD

- Extrude the text to 0.25" out of the block.
- View the created part (Figure 1.12). View → Orientation → Standard Orientation.
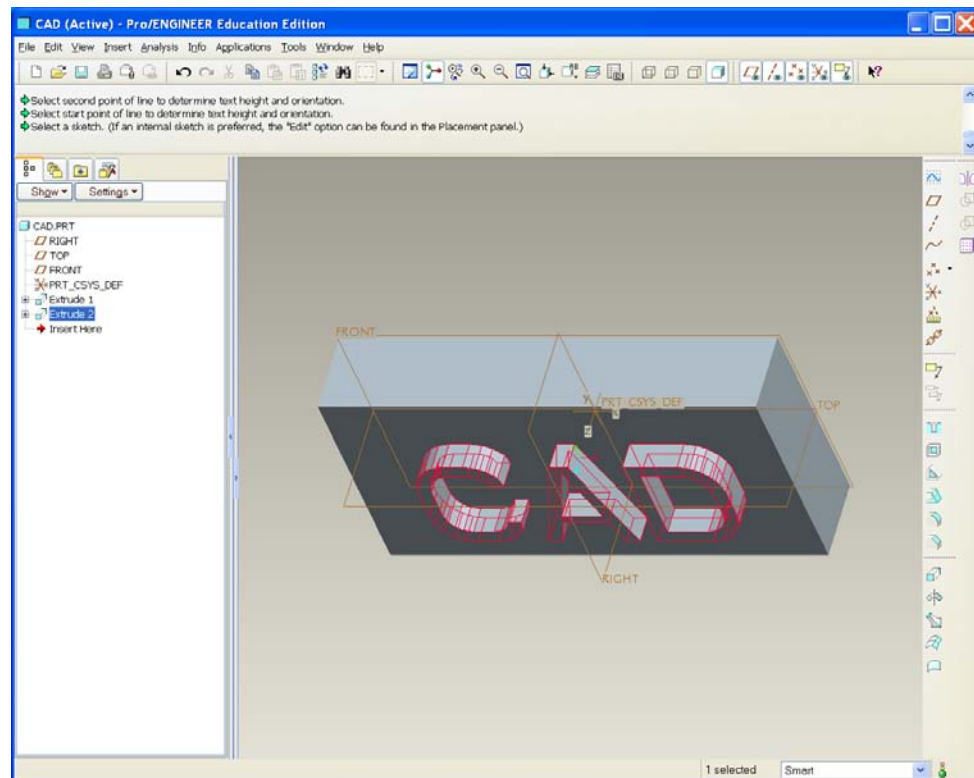- Save the part. File → Save and Exit or Close to continue.

Figure 1.12 Part model

B. Create the workpiece. The workpiece represents the raw stock of material from which the part will be machined. Pro/E refers to this procedure as an assembly operation.

1. Start the Pro/E program. Windows menu items, Start → Programs → PTC → Pro ENGINEER→ Click on the icon ![icon] Pro ENGINEER.
2. Set working directory. File → Set Working Directory, select working directory.
3. Create the part name. Pro/E main menu, File → New, select Manufacturing in Type window and NC Assembly in Sub-type window, enter name: mfgCAD as shown in Figure 1.13
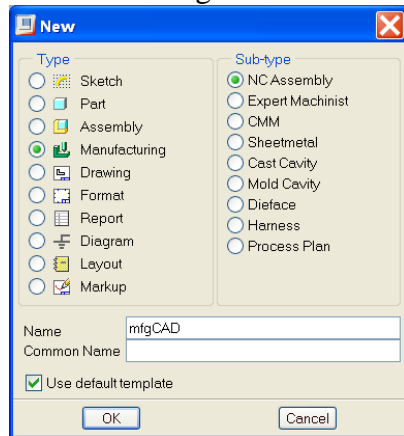
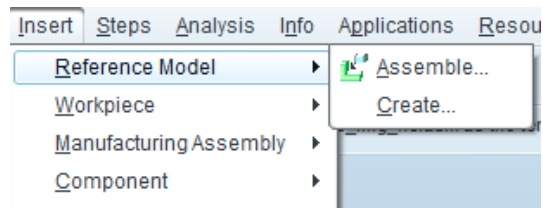Figure 1.13 Creating a new file   Figure 1.14   Figure 1.15

4. Load the part. From Insert pull down menu, select Reference Model → Assemble as shown in Figure 1.14 & 1.15. Select cad.prt in the open window.

   The component Placement window pops up. Select ![icon] to place the part at default location as shown in Figure 1.16 and click OK. Click OK to close Create Reference Model as shown in Figure 1.17.
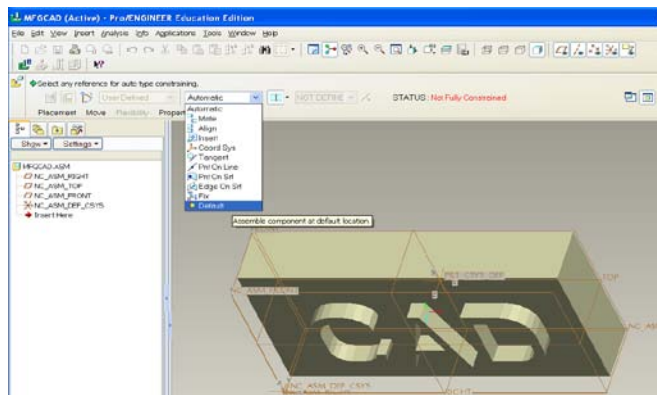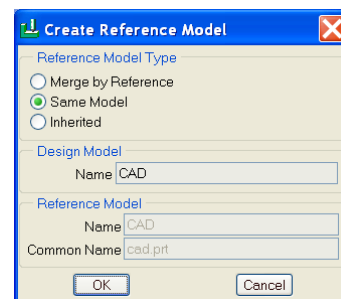
Figure 1.16   Figure 1.17

5. Create a workpiece 4"x8"x2". From Insert pull down menu, select Workpiece → Create. Enter a name for the workpiece: wpcad. Select Solid → Protrusion → Extrude → Solid → Done. Select NC_ASM_FRONT as sketch plane and NC_ASM_RIGHT as reference plane in Figure 1.18. Create a rectangle 4"x8" surrounding the CAD part as shown in Figure 1.20 with references as shown in figure 1.19 and extrude it to 2" as shown in Figure 1.21.
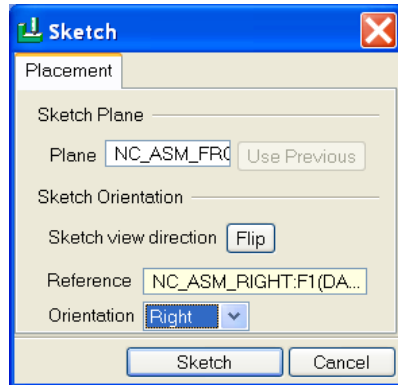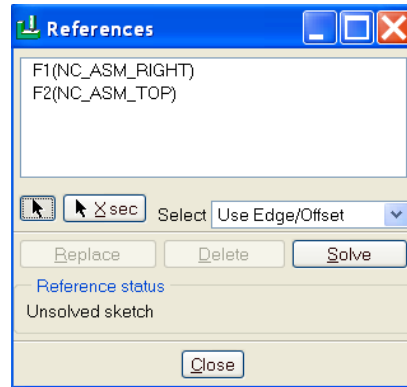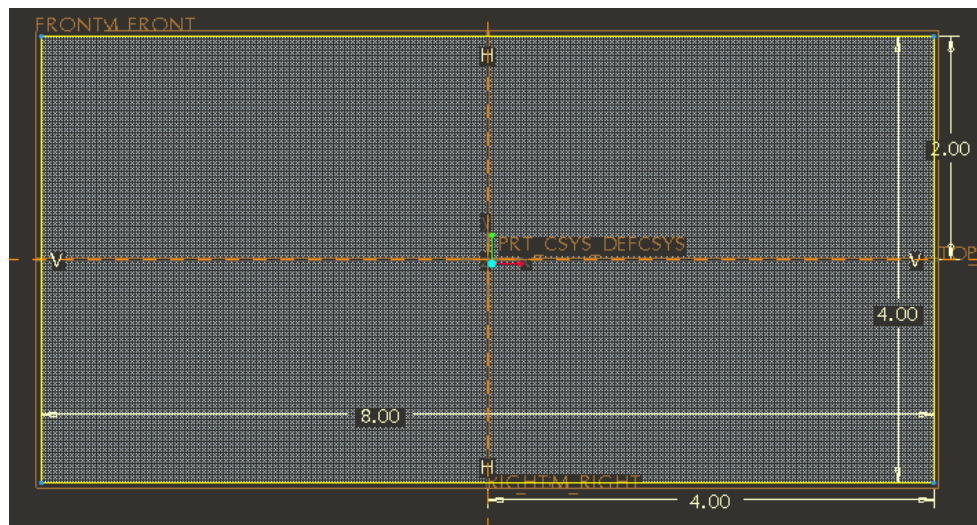
Figure 1.18
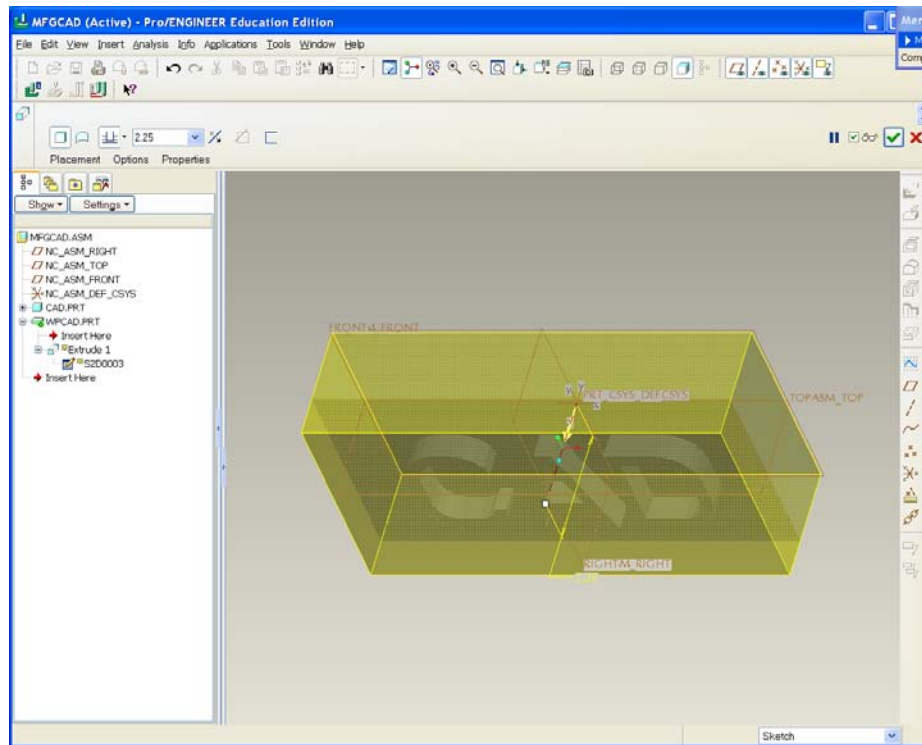
Figure 1.9

Figure 1.20

Figure 1.21 Reference Model and Workpiece

C. Perform the machining operation setup. The setup consists of defining the type of machine to use. It also requires defining a coordinate system if one does not already exist and a retraction plane for the cutting tool. The coordinate system must match the mill orientation and the part zero.

1. Insert → Model Datum → Coordinate system. Coordinate system window pop up automatically as shown in Figure 1.22.
2. Pick 3 references planes as shown in Figure 1.23 (Click two side and top planes while holding down Ctrl key) to place the coordinate system.
3. Orient X and Y axes (Figure 1.24) along the top edges as shown in Figure 1.23 by clicking the Flip button as shown in Figure 1.24.
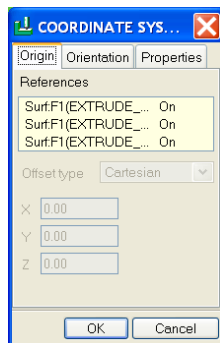
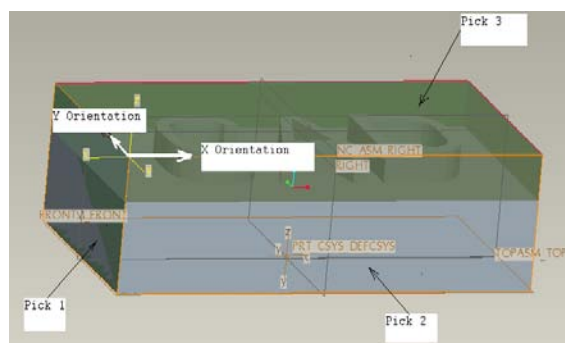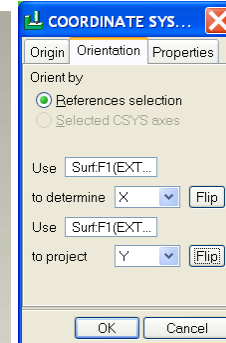

Figure 1.22   Figure 1.23   Figure 1.24

4. Steps → Operation. Operation Setup window pops up automatically as shown in Figure 1.25.

5. Define NC machine. Click NC machine icon ▦ in Figure 1.25, Machine Tool Setup windows pop up as shown in Figure 1.26. Enter the parameters as shown below

Machine name: HAAS
Machine type: Mill
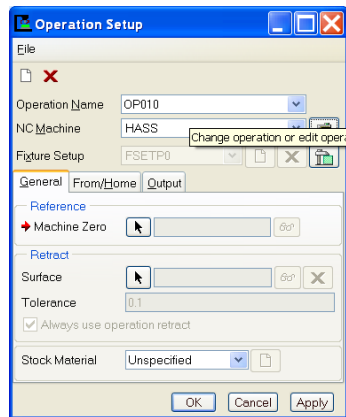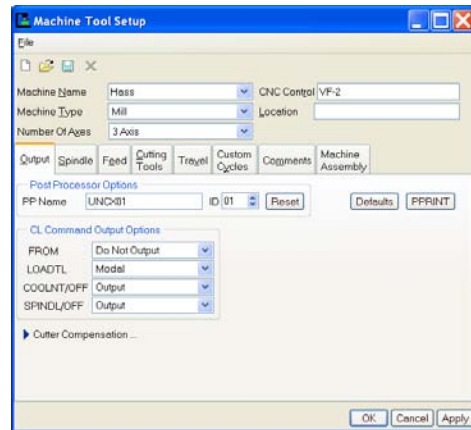Number of Axis: 3
CNC control: VF-2

Figure 1.25

Figure 1.26

6. Define Machine Zero. Click Reference Machine Zero icon [Machine Zero ▸] in Figure 1.25. Select the coordinate system ACS0 just created.

7. Define retract plane. Click Retract Surface icon [Surface ▸] in Figure 1.25. Retract Setup window pops up (Figure 1.28). Enter 0.5, click OK to close the window. The retract plane is shown in Figure 1.29.

8. Complete Operation setup. Click OK to close operation setup as Figure 1.30. MFG Setup → Done → Return.
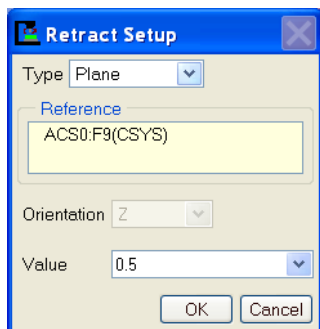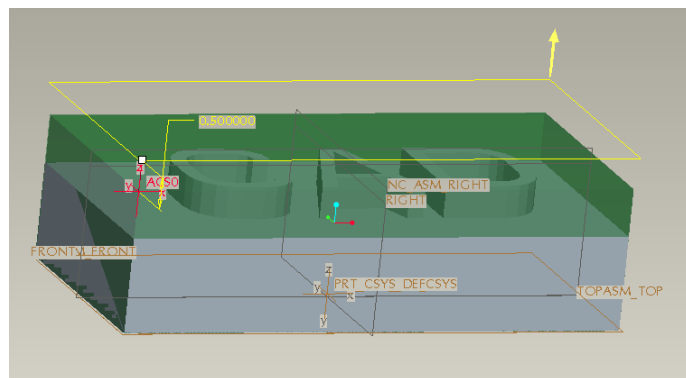

Figure 1.28

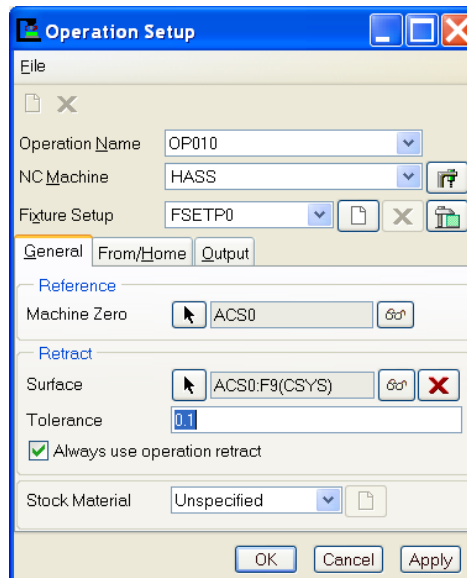Figure 1.29 Defining the Retract Surface

Figure 1.30 Completed Operation Setup

D. Define the machining operations (create mill volume). The setup consists of defining the type of tool to use and machining parameters (tools size, cutting speed, etc.), and specify the volume of material to be removed.

1. Steps → Volume Rough.
2. SEQ SETUP window pops up. Ensure that **name**, **tool, parameters** and **volume** are checked and then choose DONE. (Figure 1.31).
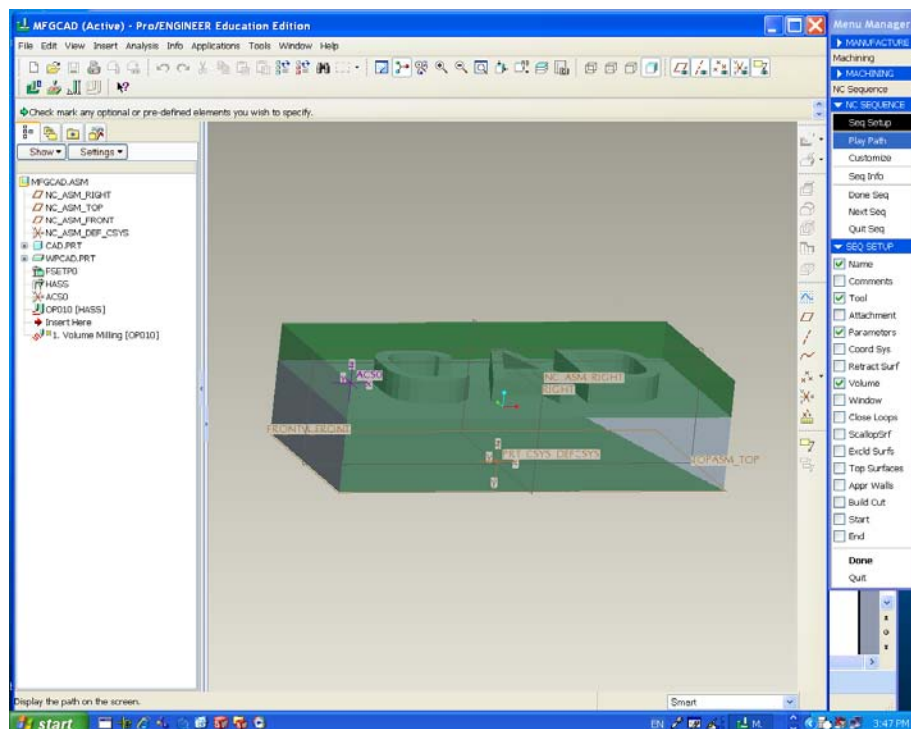

Figure 1.31

3. Enter NC sequence name: Volcut
4. Tools setup table pops up. Enter the tool values cutter diameter 0.25" and length 4" as shown in Figure 1.32 and APPLY OK.
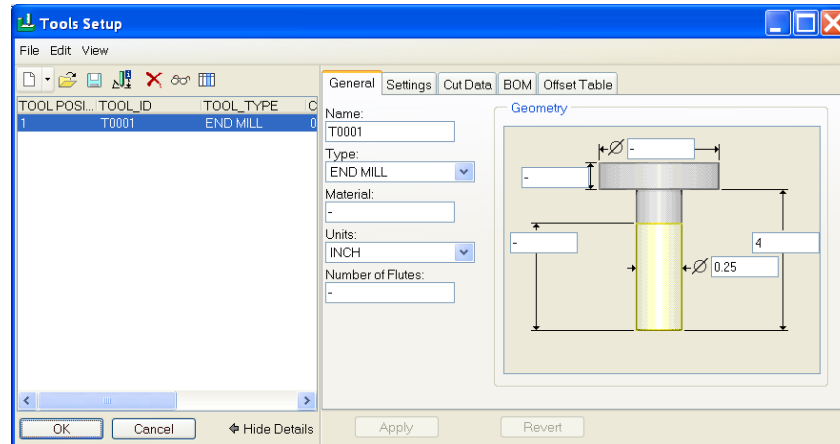

Figure 1.32

5. Edit Parameters window pops up. Input or change the values as shown in Figure 1.33. File → save as to file called "milprmVol" → click OK to exit.
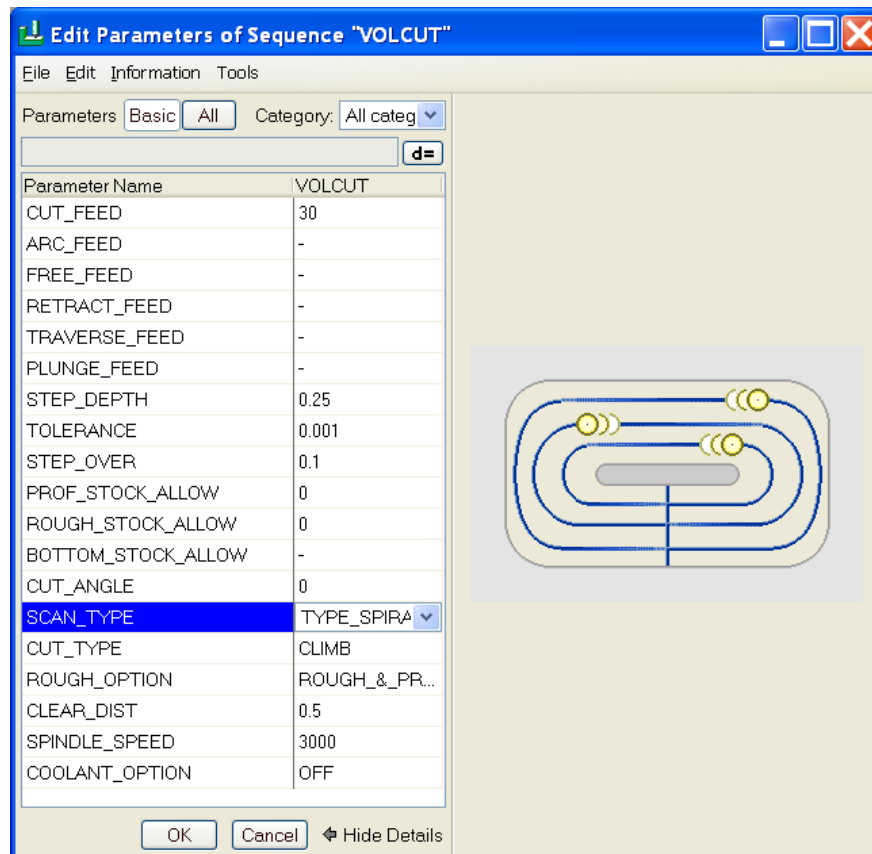

Figure 1.33

6. Select Mill Volume windows pop up as shown in figure 1.34. Since Mill Volume has not been created, there is no volume to select. Click ![Mill Volume] Tool to create the volume.
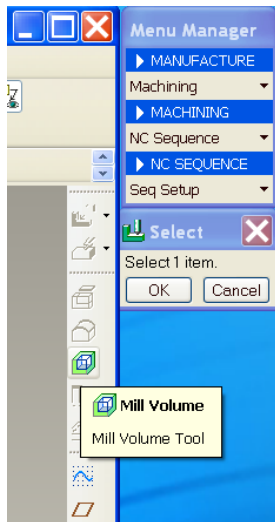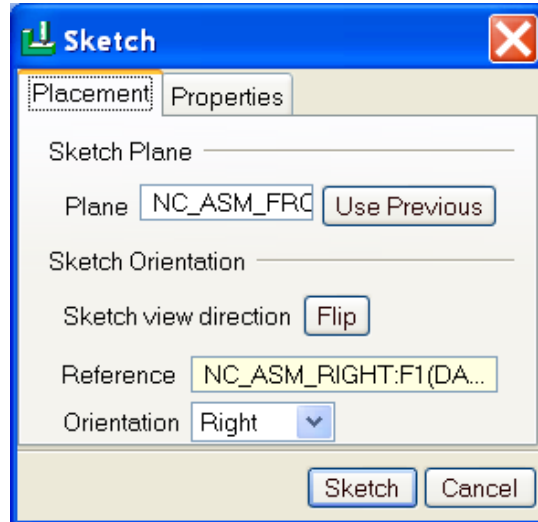

Figure 1.34   Figure 1.35

7. Create and Define a Volume. To specify the volume of material to be removed.

- Click on Sketch Tool ![icon] to sketch. Select NC_ASM_FRONT as sketch plane and NC_ASM_RIGHT as Reference plane Right as shown in Figure 1.35.

- Sketch a rectangle window 4" x 8" exactly on the top of workpiece as shown in figure 1.36. Click on ![checkmark] to exit sketch.
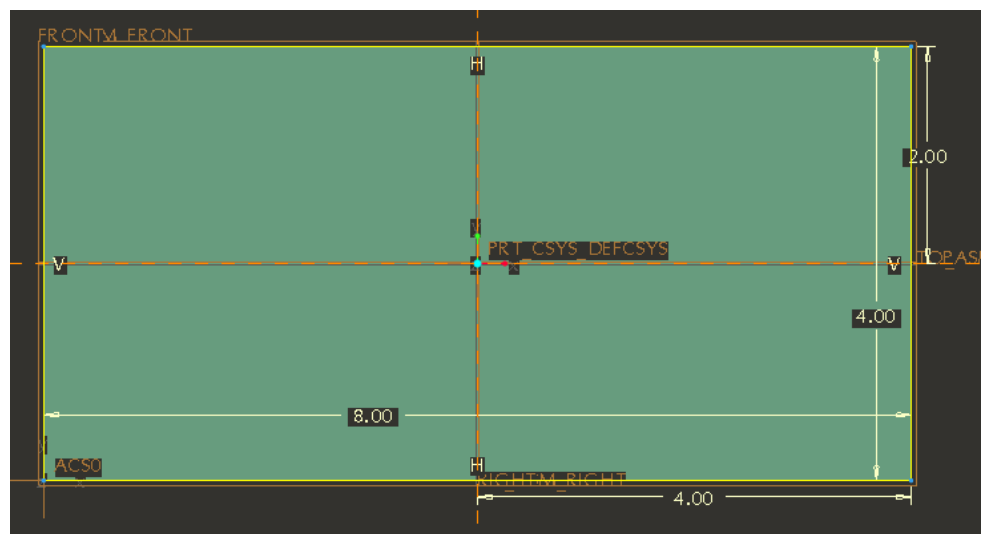

Figure 1.36

- Click on Extrude tool to extrude a rectangle to 2.25" (similar to the workpiece).

- We have sketched our entire workpiece. But we need to leave the material that represents our part. At this point Pro/E provides a Trim function that will "trim" the part from mill volume. Select **Trim** and select the part we wish to be "trimmed" out of the mill volume as shown in Figure1.37. We have defined the volume to be removed (the workpiece minus the part).
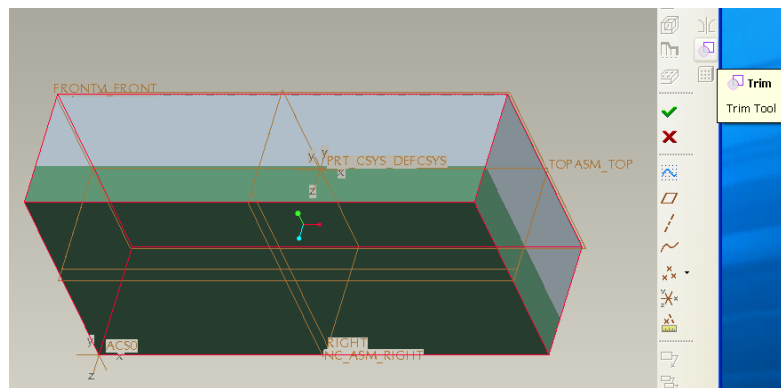

Figure 1.37

8. View tool path. Menu Manager → NC Sequence → Play path → NC Check. The cutting path is shown in Figure 1.38.
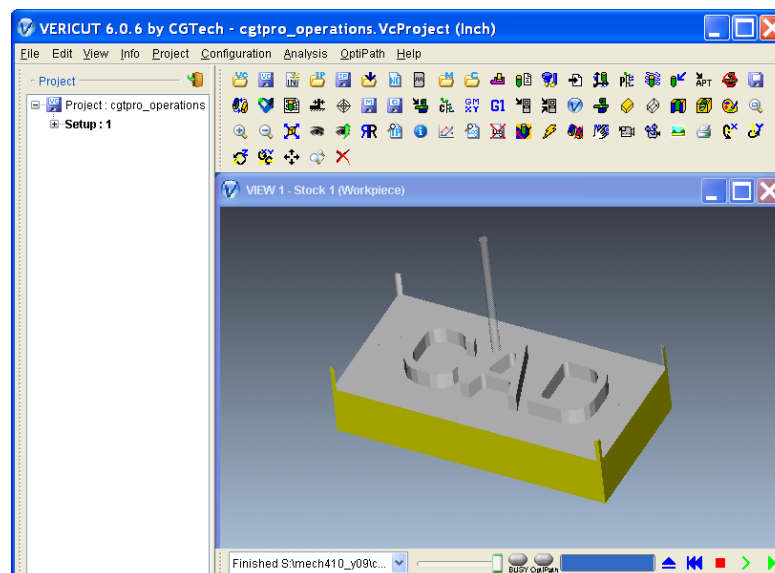

Figure 1.38

9. After this completes **choose DONE SEQ. If you don't do this, you are likely to lose the definition of this NC sequence.**

E. Define the machining operations (create mill windows). The setup consists of defining the type of tool to use and machining parameters (tools size, cutting speed, etc.), and specify the volume of material to be removed.

1. Steps → Volume Rough.
2. SEQ SETUP window pops up. Ensure that **name, tool, parameters**, and **window** are checked and then choose DONE. (Figure 1.39).
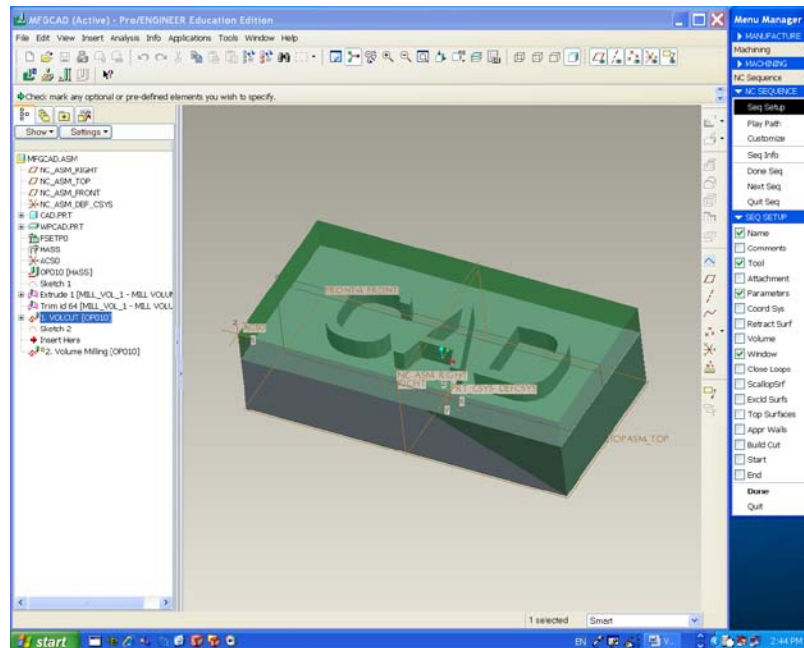

Figure 1.39

3. Enter NC sequence name: Wincut
4. Tool setup table pops up. Click ok to use tool 1 as previous tool (see Figure 1.32).
5. Edit parameters window pop up. File → open → select milprmVol → open → Ok (see figure 1.33).
6. Select Wind window pop up as shown in figure 1.40. Since a windows had not been created, there is no windows to select. Click on Mill Windows

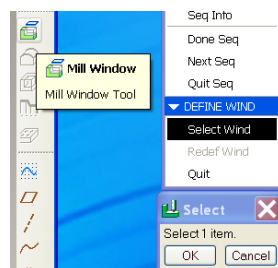   Tool to create the window as shown in figure 1.41.
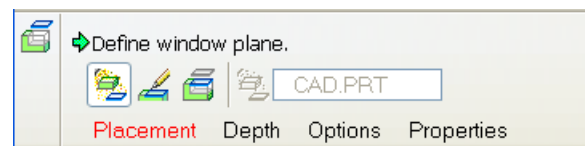

Figure 1.40   Figure 1.41

7. Create and define a Mill Window. To specify the volume of material to be removed. We defined a "mill windows" on the top surfaceof our workpiece that include the entire workpiece. Think of the workpiece was being transparent and the surface as being solid. If you look down through the mill windows, you will see all the volume of the material in the workpiece located around the surface.

- Click on the Sketch window type ![icon] to define the window plane and sketch the window. Select top surface as sketch plane and click on the define an Internal sketch to sketch as shown in Figure 1.42.
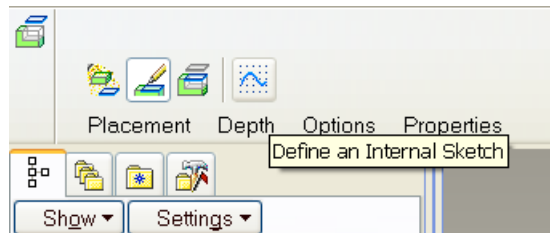

Figure 1.42

- Sketch Window pops up. Select NC_ASM_RIGHT as reference RIGHT as shown in Figure 1.43. Click on Sketch.
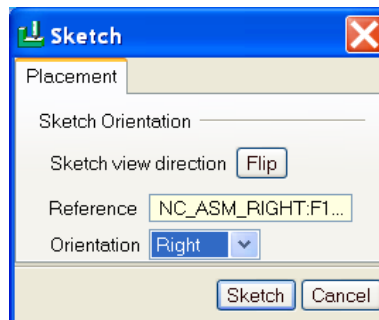

Figure 1.43

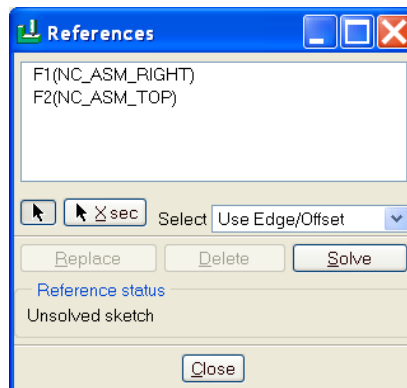- References Window pop up. Select two planes as shown in Figure 1.44.

Figure 1.44

- Sketch a rectangle window 4" x 8" exactly on the top of the workpiece as shown in Figure 1.45. Click on ✔. Click on Options and choose ON Window Contour as shown in Figure 1.46 and Click ✔.
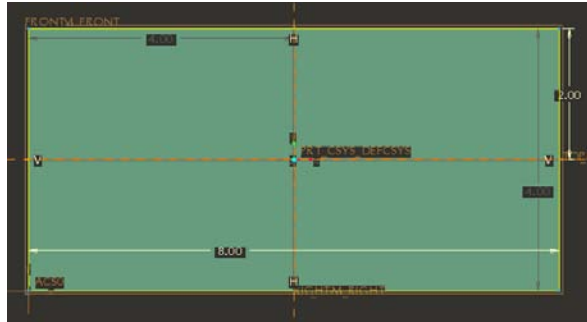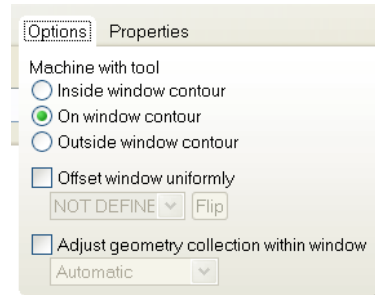

| Figure 1.45 | Figure 1.46 |

8. View tool path. Menu Manager → NC Sequence → Play path → NC Check. The cutting path is shown in Figure 1.47.
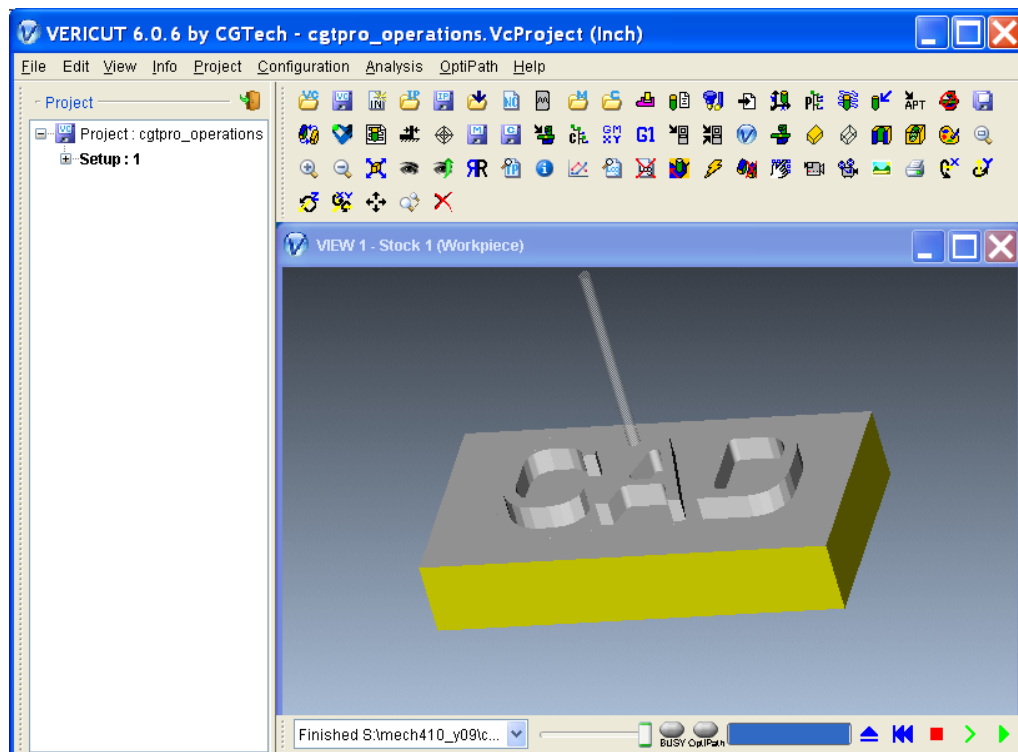

Figure 1.47

9. After this completes **Choose DONE SEQ. If you don't do this, you are likely to lose the definition of this NC sequence.**

F. Create cutter location file and NC code.

1. Edit → CL Data → Output → Select Feature → NC Sequence → volcut. PATH → FILE → Done. Enter name: volcut in the Save a Copy window. Pro/E will save the file as volcut.ncl.1.
2. The cutter location file can be converted to G-code file through Pro/E post processor for HAAS VF-2 CNC milling machine.
3. Tools → **Post Process** on the CL DATA menu. Select the file volcut.ncl from the Open Dialog box and select Open to close the window. In the PP Options menu make sure the Verbose and Trace option are selected.
4. Post processing is the act of converting the toolpaths from a standard language file, called a cutter location file (**.ncl), to the language of our specific CNC machine controller, i.e. G-codes.
5. Select **Done** on the PP OPTIONS menu and a list of post processors appears.
6. Choose the UNCX01.p15 Option.
7. A file with extension ".tap", i.e., volcut.tap is created, which contain the G-code.
8. Close the information window and Select DONE/RETURN from CL DATA menu.
9. Open the volcut.tap file to view using program WORDPAD.

The volcut.tap file is shown below:
```
%
N0005(FADAL VMC 6030 - VH65)
N0010G90G40G80
N0015T1M6
N0020S3000M3
N0025G0X6.0282Y2.025
N0030G43Z.5H1
N0035G1Z-.25F30.
N0040X1.9782
N0045Y1.975
..............
N4815X3.989Y1.8059
N4820X4.3944
N4825X4.1917Y2.4269
N4830X3.989Y1.8059
N4835X4.1268
N4840Z.5
N4845M5
N4850M2
%
```