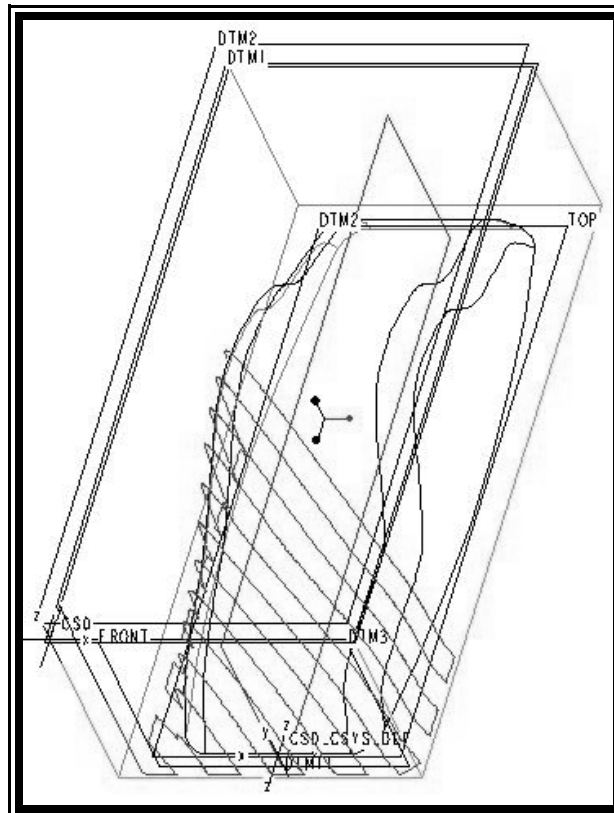


A Pro/Manufacturing Tutorial

Pro/ENGINEER - Wildfire 2.0



Paul E. Funk
University of Evansville
Evansville, IN

Loren Begly, Jr.
Whirlpool Corporation
Evansville, IN

SDC
PUBLICATIONS

Schroff Development Corporation

www.schroff.com
www.schroff-europe.com

Visit the following websites to learn more about this book:



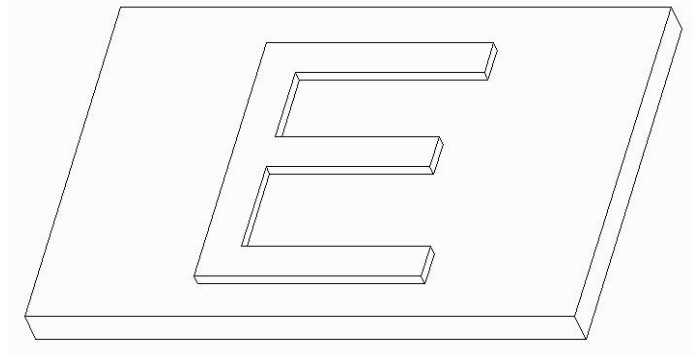
[amazon.com](https://www.amazon.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

Section 1 -- A Quick Run Through

In this section we'll develop the CNC code for a 3 axis CNC mill to produce a simple rectangular block with a raised letter.

**Figure 1-1
Part for Section 1**



Pro/Manufacturing can quickly become very involved and complex. This exercise is designed to work through a single manufacturing sequence to familiarize the user with the general procedure involved. Although a single manufacturing sequence (volume milling), it still involves a lot of commands. Since we're just trying to gain familiarity with the general procedure, we'll present this example with a minimum of explanation. At the beginning of each step we'll explain what we're going to do, then we'll do it and at the end of the step we'll tell you what we (and you) did. Later sections will go into detail about why you made the choices you were told to make and what alternatives you have available to you.

Produce the Part Model

We'll follow the steps discussed in the preface. First, 1) Produce the part model. Before entering the Manufacturing menu, enter the Part menu and create the block shown above. A click-by-click procedure is included in the Appendix A and B. You can name the part whatever you wish, but we'll refer to it as "block" throughout the remainder of this section. Save the part and

Window

Close

File

New

○ **Manufacturing** (Leave NC Assembly checked.)

Enter the name you want to use for your manufacturing process. We'll use "block". (Pro/E will add a .mfg extension.) Select

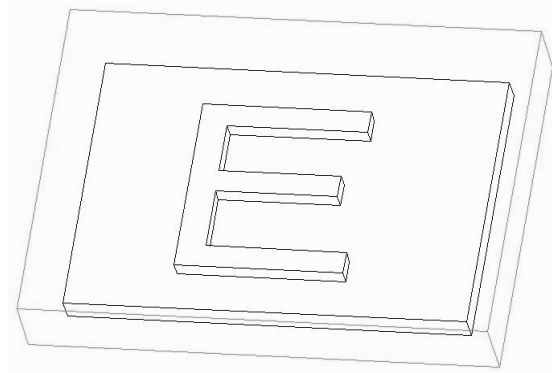
OK

That completes step 1).

Assemble the Part and Workpiece

We're now ready for step 2). We'll create the workpiece material the part will be machined from using our part as a guide. We'll make the workpiece bigger than our original part.

**Figure 1-2
Part/Workpiece Assembly**



We begin by selecting

Mfg Model

Assemble

Ref Model

and pick on the part name for the part we want to manufacture (block.prt).

Open

Pro/E displays the part. We'll use this part as a guide to create the workpiece from which the part will be machined.



(Assemble component at default location.)

OK

Create

Workpiece

We gave it the name, block_wp. We're now going to enter Sketcher, just as if we were building a part in part mode, to model the workpiece. Select

Protrusion

Done

Placement

Define

and select the sketching plane. Use the right mouse button to highlight the bottom surface of the block and the left mouse button to select it. (Refer to Figure A if you don't recall which is the bottom surface). Pick on the front surface (Refer to Figure A) of the block for the Reference and select

Bottom

for the Orientation. Select

Sketch

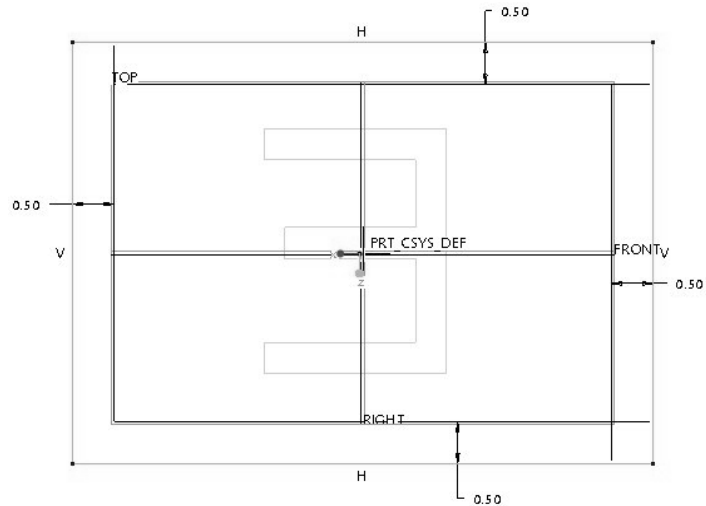
Select any two perpendicular edges as references. The actual references are not important the way we'll dimension the section. Select

Close



and create a rectangular section as the outline for the workpiece so that it completely encompasses the outline of our block. Dimension the rectangle you just created to be ½ inch bigger than block.prt on each side.

**Figure 1-3
Dimensioning Scheme**



Select



Go to the default view and make sure that the protrusion encompasses the part. If it doesn't drag the depth handle or use



to change the depth direction. Enter 1.0 inch as the depth of the protrusion. Select



Done/Return

and we've completed the creation of our workpiece. In default view the part/workpiece assembly should appear as shown in Figure 1-2.

We've completed step 2). We have our original part model assembled within the workpiece from which it is to be machined.

Manufacturing Setup

We're now ready to 3) perform the manufacturing setup. We will specify a 3 axis milling operation, create a coordinate system for our operation and create the volume of material to remove by milling. Select

Mfg Setup

Pro/E displays the Operation Setup dialogue box. Click on the “mill icon” at the far right of the NC Machine box.



Make sure the Machine Type is “Mill” and the Number of Axes is “3 Axis”. Select

OK

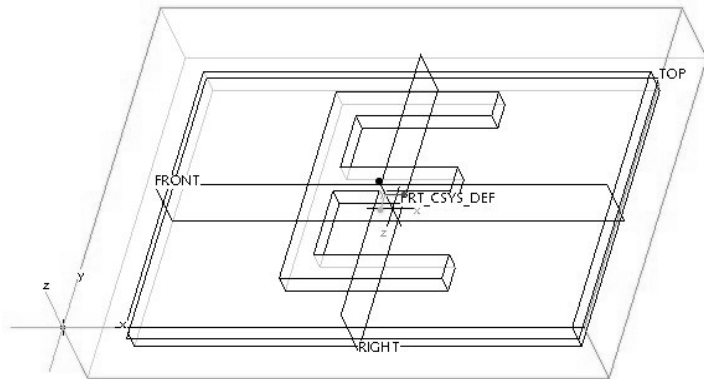
and click on the arrow next to Machine Zero. Select

Create

and pick on the workpiece. Pro/Manufacturing displays a coordinate system dialogue box.

We'll place a coordinate system at the intersection of the top-left edge of the workpiece with the top front edge. Click on the **top-front** edge to indicate the location of the first reference. Hold down the control key and click on the **top-left** edge to indicate the second reference for location of the coordinate system.

Figure 1-4
Pro/ E Axis Display



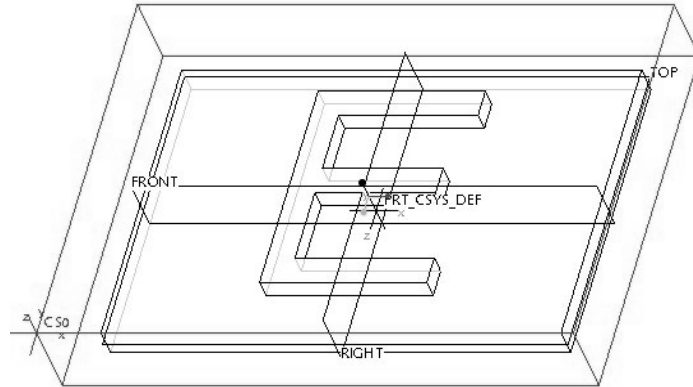
Click on the *Orientation* Tab. If necessary, flip the x axis so that it extends to the right and flip the y axis so that it extends from the front to the back of the workpiece (Figure A).

Select

OK

OK

**Figure 1-5
Part Home**



To complete the manufacturing setup, we still need to specify the volume of material to be removed. We'll select the entire workpiece and the "remove" the part from this volume. Select

Mfg Geom

Mill Volume

Create

and enter a name. We'll use "mv1". We'll use the sketch command to create the volume of material to be removed from our workpiece. We'll remove all the workpiece material that lies outside the part. Select

Sketch

Done

Done

Again use the right mouse button to highlight the bottom surface of the workpiece as the sketching plane and the left mouse button to select it. Make sure the protrusion arrow extends from the bottom of the workpiece upward and select **Okay**. Select **Bottom** from the SKET

VIEW menu and pick on the front surface of the workpiece. Again select two perpendicular edges as references. Select

Close

Sketch

Edge

Use

and select all four outer edges of the workpiece (not the part). Select

Close



Up to Surface

Done

and pick the top (Figure A) surface of the workpiece. (You may want to use the default view.)

Select

Ok

We've selected our entire workpiece as the mill volume. 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 the mill volume. Select

Trim

and use the right mouse button to select the part we wish to be "trimmed" out of the mill volume.

Use the left mouse button to accept the selection.

Done/Return

Done/Return

Done/Return

and we've defined the volume to be removed (the workpiece minus our part).

And we've completed the manufacturing setup. We defined a 3-axis milling operation, created a coordinate system for our workpiece, and created a mill volume representing the material that is to be removed by machining.

Machining Sequence

Let's 4) define the machining operations. We'll select a volume milling sequence, define our tool and machining parameters (tool size, cutting speed, etc.), create a retraction plane and specify the volume of material to be removed (created in the previous step). Select

Machining

NC Sequence

Done (Leave Volume and 3 Axis highlighted)

Done (notice the checked parameters we must define)

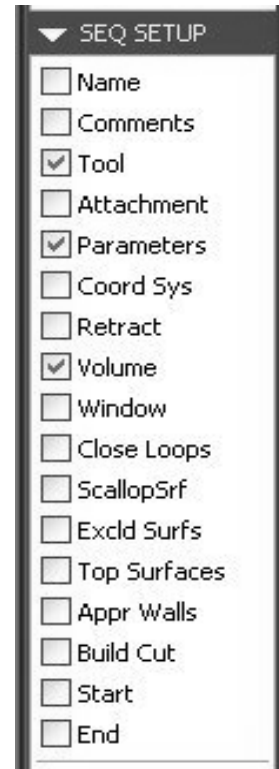
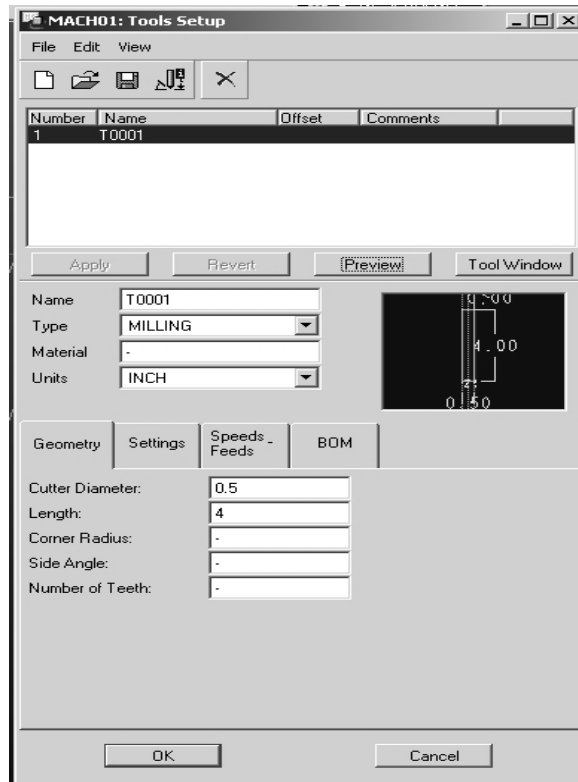


Figure 1-6
SetUp Parameters

Pro/E will display a Tool Setup Table.

Figure 1-7
Tool Setup Table



While the user can modify any of the values, we'll just specify the diameter and length for this example. Click on the specified parameter and change to the following values:

Cutter_Diam .25

Length 2

To save the values and exit the Setup table,

Apply

OK

Set

and Pro/E will display the remaining machining parameters. While the user can modify any of the values, all parameters that have a value of -1 must be specified.

**Figure 1-8
Machining
Parameters**

Manufacturing Parameters	Volume Milling
CUT_FEED	-1
STEP_DEPTH	-1
STEP_OVER	-1
PROF_STOCK_ALLOW	0
ROUGH_STOCK_ALLOW	0
BOTTOM_STOCK_ALLOW	-
CUT_ANGLE	0
SCAN_TYPE	TYPE_3
ROUGH_OPTION	ROUGH_ONLY
SPINDLE_SPEED	-1
COOLANT_OPTION	OFF
CLEAR_DIST	-1

Use the following values:

Cut_Feed 60

Step_Depth .125

Step_Over .125

Scan_type Type_Spiral (We'll explain why you change this later.)

Spindle_Speed 1000

Clear_Dist 1

After entering all the above values, select

File

Exit

to save the values and exit the table. Select

Done

Pro/E now prompts us to create a retraction plane; ie, a plane to determine the height the tool will withdraw to each time it retracts from the workpiece. We'll put a retraction plane one inch above the part (0.5 inches above our coordinate system). Select

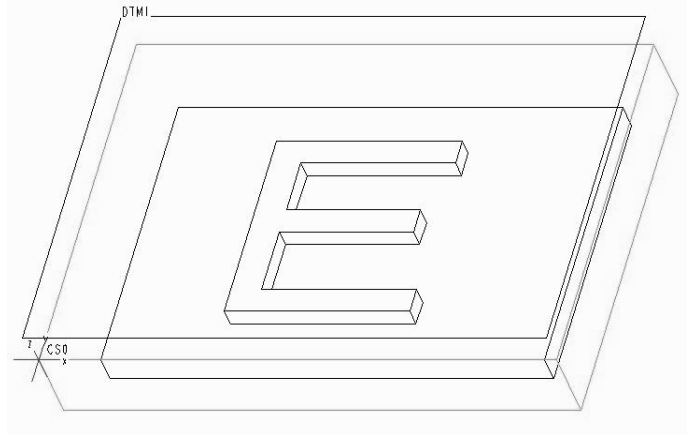
Along Z-Axis

and enter a value of 0.5 (1/2 " above the workpiece).

OK

Pro/E displays the retraction plane above the workpiece.

**Figure 1-9
Retraction Plane**



Now we must specify the volume of material to be machined. Since we've already created the volume (recall we called it mv1), all we have to do is select it. Select

Select Vol

and pick on the mill volume created earlier in this section. We've finished the process.

And we've finished defining our machining sequence. We defined a volume milling sequence, entered tooling and machining parameters, created a retraction plane and selected the volume of material to be removed by milling.

Viewing and Outputting Results

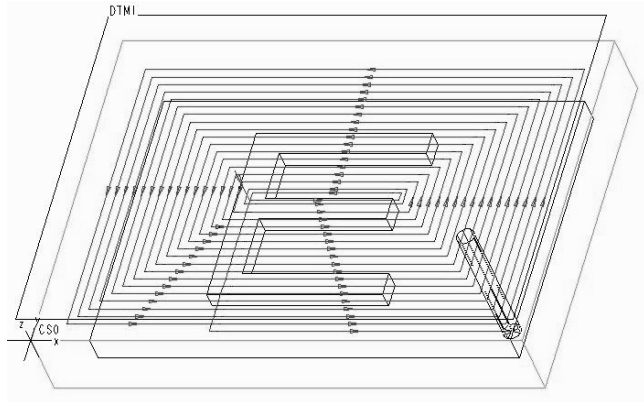
Although we're done, at this point we need some "proof" that everything we've done is okay. We can get that proof by creating the tool path (CL - cutter location) and viewing the cutter location as it removes the material. We'll "play the path" of the tool. Select

Play Path

Screen Play

and Pro/E plays a radio control, Play Path window. And if you've done everything correctly, you can use the radio buttons to display the cutting tool's path centerline as it removes the material you've indicated to be mvl.

Figure 1-10
Cutting Path Display



After you've played the tool path (CL data), you can also run an NC check to graphically depict the material removal. Close the Play Path dialogue box and select

NC Check

and use Vericut controls (the green button on the bottom right of the screen) to watch as Pro/E simulates how the material is removed.

Figure 1-11
NC Check



That's everything involved in the process except posting or postprocessing the CL data to generate the G-code. That's pretty straight forward and basically just requires a listing of menu picks. We'll save that as part of a later exercise.