Creo 2.0, ND Logo CAM Tutorial
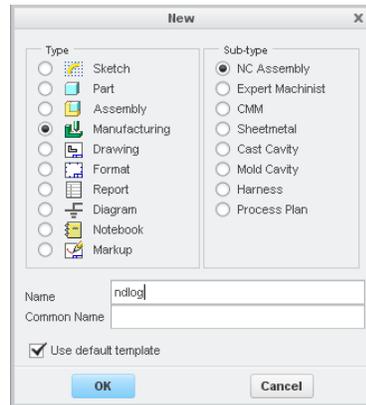

In this tutorial we will machine the monogrammed ND emblem, as shown above.

To begin, start Creo Parametric and set your working directory to the CAM_tutorials folder.

Create a Manufacturing Model

Step 1:   Select File >> New
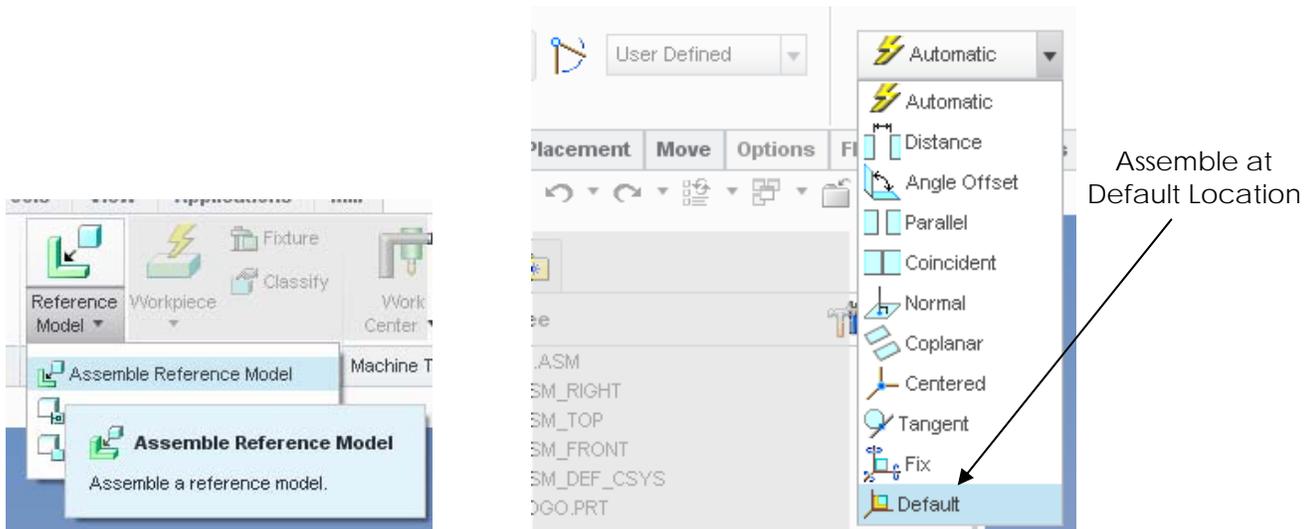
Step 2:   Select Manufacturing Type on the New dialog box, NC ASSEMBLY as the Sub-type, Name >> ndlogo

Define a Reference Model

Step 1:   In the Manufacturing tab select >> the icon or Reference Model >> Assemble Reference Model

Step 2:   Open nd_logo.prt as the part to manufacture.
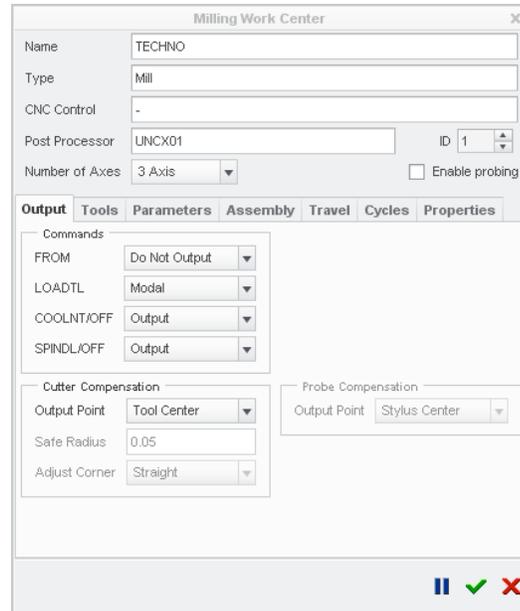

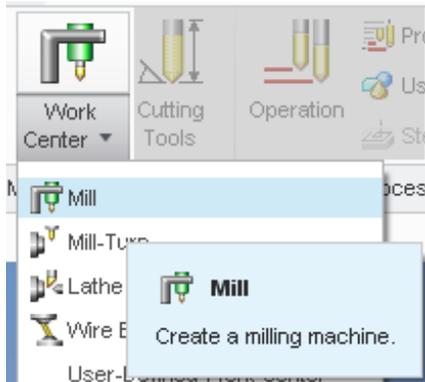Assemble at Default Location

Step 3:   Use the **Assemble Component at Default Location** Constraint Type, constrain the location of the reference model.

Step 4:   Select to exit the Component Placement menu.

Create a Work Center

Step 1: In the Machine Tool Setup area, Select >> **Work Center >> Mill** to open the **Milling Work Center** dialog box.
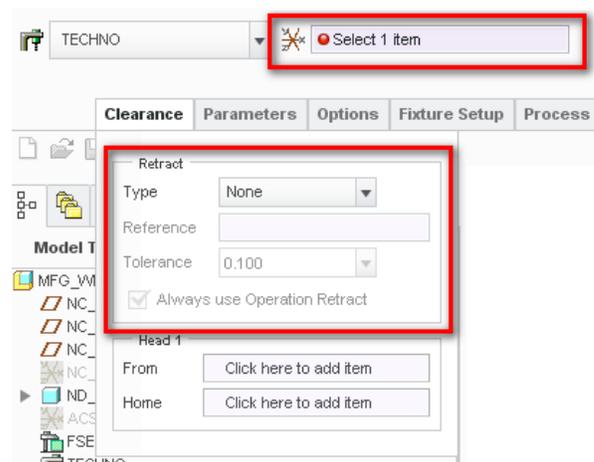
Step 2: In the Machine Work Center dialog box, change the machine Name to **Techno**. Make sure the Type is set to **Mill** and the Number of Axes is set to **3**.

Step 3: Select ☑ to exit the Milling Work Center menu.

Define an Operation

Step 1: In the Process area, Select >> Create a New Operation to open the Operation Setup dialog box.

Step 2:   You are immediately prompted to select a coordinate system for this operation.

The positive (+) Z-Axis of your coordinate system MUST point upward, normal to the top surface of your reference part!!   Note: It is common to define your machine zero in a corner (or center), on the top surface of your material, since this is easy to locate during setup.

Step 3:   Select the coordinate system, NC_CS_Machine on the model.
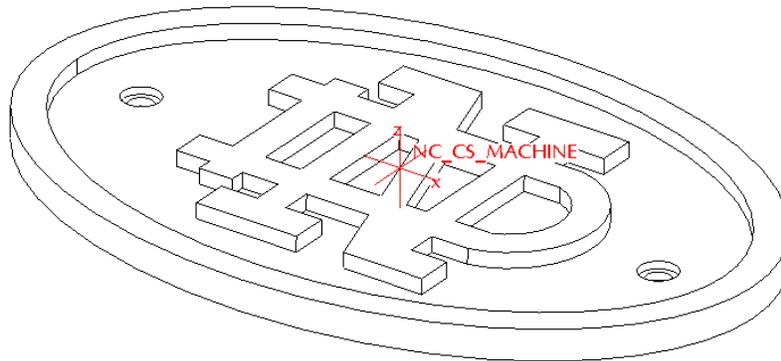
Step 4:   Select the Clearance Tab to setup the Retract Plane information.
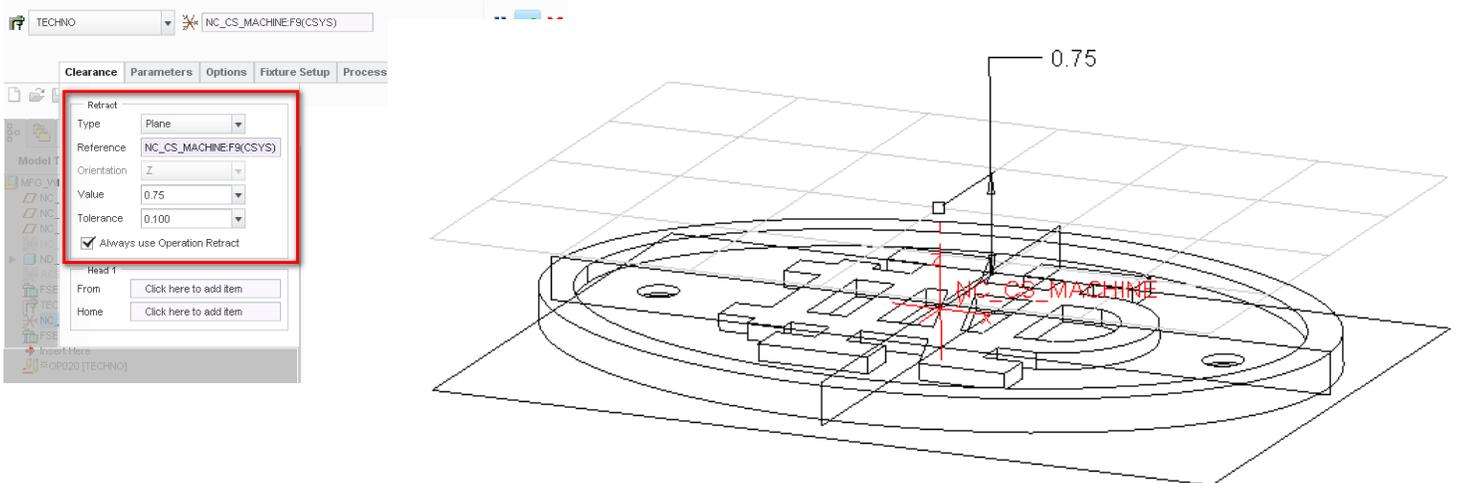
The retract surface defines the level to which the tool is retracted after a cut.

**Step 5:**   Under Type, select Plane and for a Reference, select the NC_CS_Machine coordinate system.

Step 6:   Enter .75 as the Value.

Step 7:   Select ✅ to exit the Operation Setup menu.

Step 8:   Save your manufacturing model.

Volume Milling

Within this segment of the tutorial you will define a milling sequence to remove the material around the logo shown below. The volume milling parameters that you define include; the cutting tool (1/8 inch end mill), the manufacturing parameters, and the volume to mill. You will define the volume to mill using, extrude and trim to part, for this milling sequence.

Volume to remove
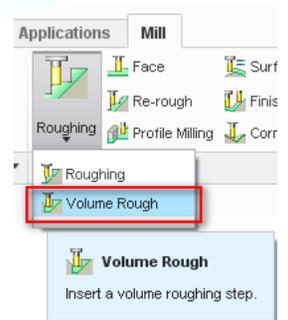
Step 1:   Select the Mill (Tab) >> Roughing >> Volume Rough from the toolbar.

Step 2:   The Sequence Set-up Dialog Box appears as shown on the right.

Note that WINDOW is pre-selected.

Step 3:   Select >> VOLUME (Tool and Parameters)

Creo automatically checks the minimum parameters required for a specific machining operation. Many of these selections, such as name and comments, are optional. Others, such as the tool and parameters, are required. Starting from the top of this list (Tool in this case), Creo will automatically move you through the order of the required menus and dialog box to define each parameter.
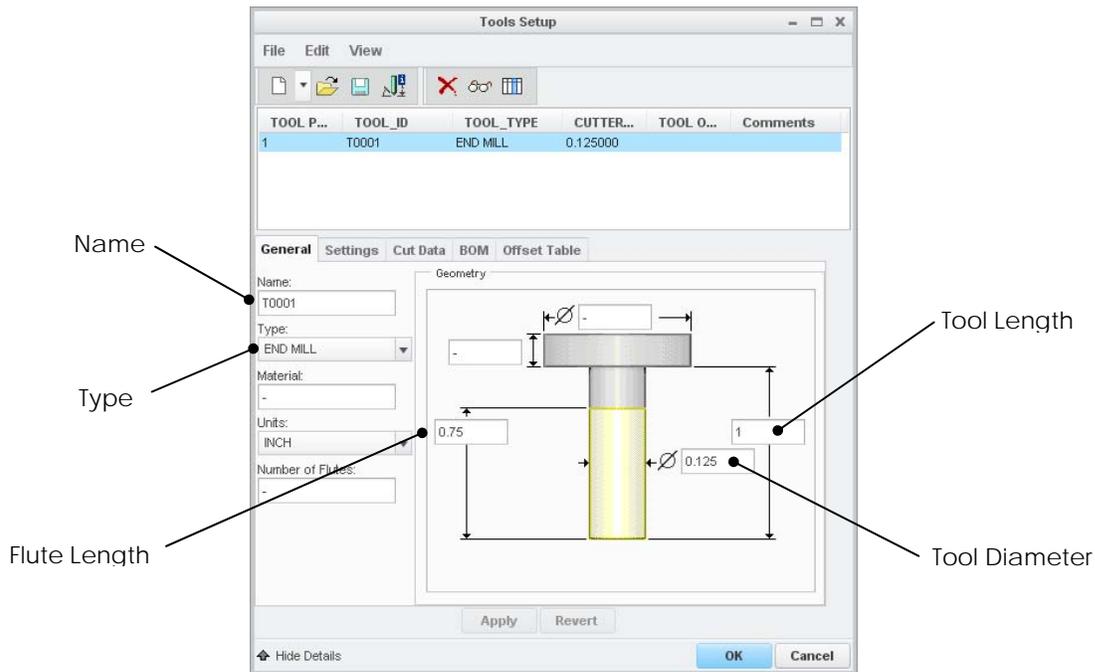
Step 4:   Select DONE on the Sequence Setup menu.

Next, the manufacturing mode will launch the Tool Setup dialog box.

Step 5:   Enter the tool parameters shown below.

- **Name:**   Defines the specific tool number (i.e. T0001).
- **Type:**   Set End Mill as the type of tool.
- **Diameter:**   Sets the diameter of the cutting tool (i.e. 0.125).
- **Flute Length:** Defines the flute length of the cutting tool (i.e. 0.75).
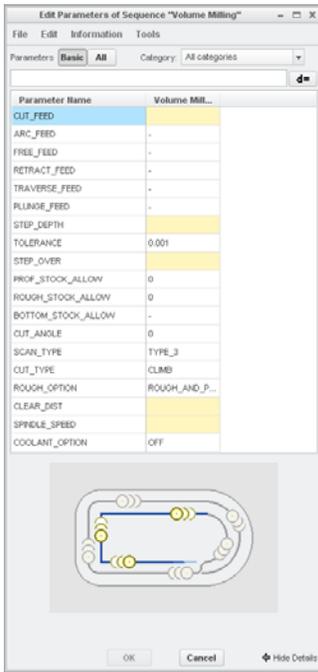

Step 6:   Select APPLY to create the tool, then Select OK to close the Tool Setup box.

The next several steps will define machining parameters for the tool selected for this particular machining sequence.

Note: The parameters shown as a Blank or Yellow Cell need to be filled in.

The general types of manufacturing parameters are:

- Feeds and Speeds—Acceleration and Feed parameters.
- Cut Depth and Step Over—Cut parameters, such as step depth, step over.
- Cutting Motions—Parameters that define the type of the cut, such as scan type, cycle type.
- Entry/Exit Motions—Parameters that define the entry and exit path for the tool, such as plunge angle, lead-in, approach and exit path.
- Machine Settings—Machine-related parameters, such as speeds, registers, coolant.
- General—Machine and NC data file names.

```
CUT_FEED          60



STEP_DEPTH      .0625

STEP_OVER       .0625




CLEAR_DIST      .2
SPINDLE_SPEED  6200
```

Step 7:   Enter the manufacturing parameters shown above.

Step 8:   Select OK on the Manufacturing Parameters menu.

The next few steps … defining the mill volume

Step 9:   Select the Mill Volume Tool icon and then select the extrude tool.

You will define this mill volume by sketching an ellipse and extruding the section the depth of the pocket (0.125 inch). The actual mill volume will be defined by trimming empty space from this sketched volume.

Step 10:   Select the sketching plane shown below, the top surface of the part. Orient the part to match the sketching environment shown below.

Sketching Plane

Ry 1.85
Rx 2.85
Outer Edge
Reference Edge
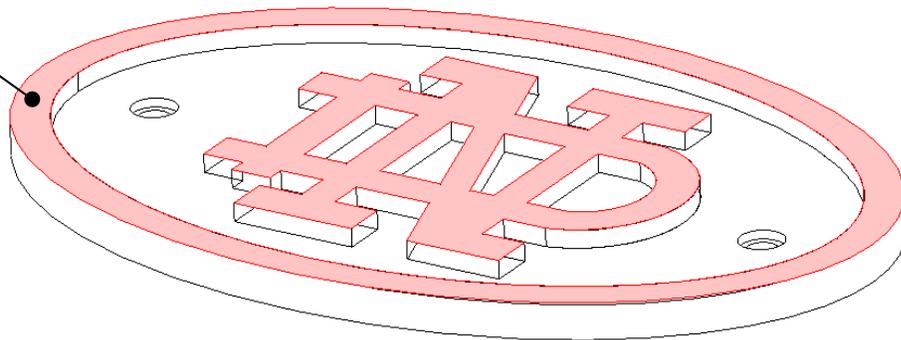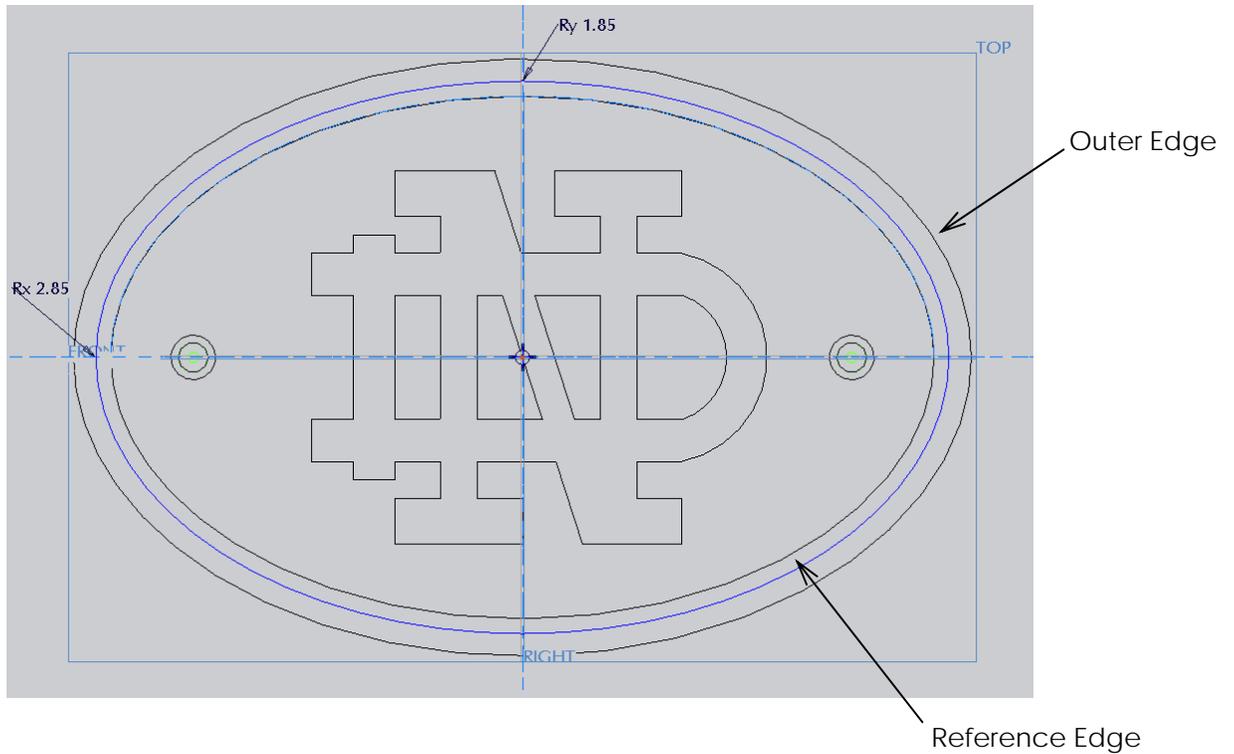
Step 11: Sketch an ellipse around the volume to be milled as shown above.
Note: the actual dimensions of the sketched ellipse are not critical as long they are the larger than the reference ellipse and smaller than the outer

Step 12: Modify the dimensions of your sketched ellipse to fall between the Reference and Outer edges.

Step 13: Select ✔ to exit the sketching environment.

Step 14: Extrude the "ellipse" section a BLIND distance of 0.125 inch or use the ⟂ ▾ UP TO SURFACE depth option and select the "yellow surface". Make sure the extrude arrow points into the part!
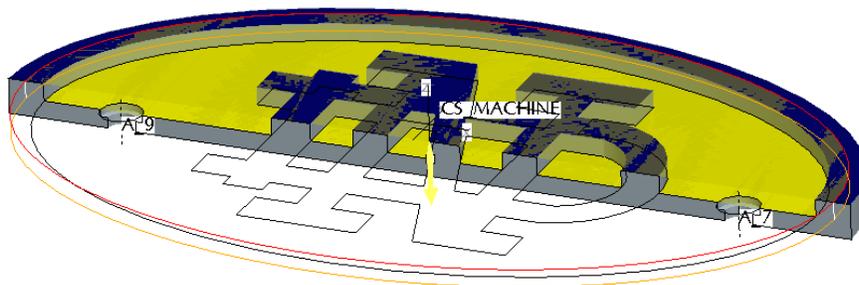
Step 15: Select ✔ to apply and make changes.

8

Step 16: Select the TRIM icon ⬚ Trim and select the part model to trim by.
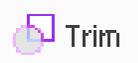
The Trim option will define the mill volume by subtracting the model from the extruded sketch.

Step 17: Select ✔ to apply and make changes.

Note: Before you verify the Toolpath … hide the "Extrude" for the Mill Volume in the model tree. This will also be useful for the following sequence.
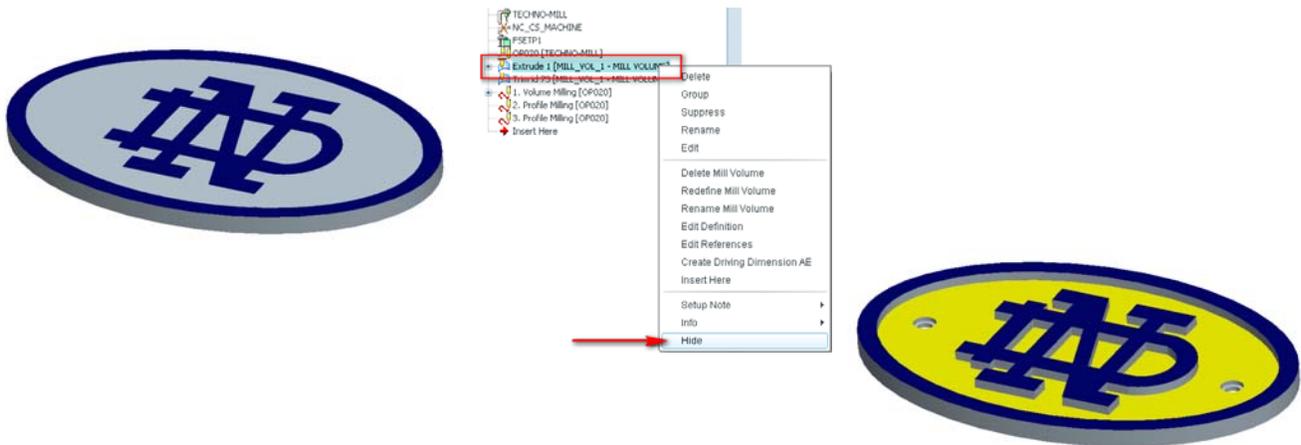
Step 18: Select PLAY PATH >> SCREEN PLAY.

The speed of the tool path verification can be slowed down using the Display Speed option.

Step 19: Close the Play Path dialog box.

Speed Option

Step 20: Select **DONE SEQ** on the NC Sequence menu.

Step 21: Save your manufacturing model.

Using a Profile Sequence for Hole Making

Within this segment of the tutorial, you will create an NC sequence that will machine the countersunk holes using a Profiling Sequence. While there is a hole-making sequence in Creo, there are several reasons why we will use a profile sequence. First, drilling holes requires a tool which is "exactly" the size of the hole to be machined. When drilling a hole, the machine simply plunges the spinning tool into the workpiece. So, whatever size tool is in the machine is the size of the hole produced. We can not stock the tooling required to drill all hole sizes! Note: we have a countersunk hole which requires a special stepped drill! Second, most of the bits available are 2-flute flat end-mills and are not well suited for drilling. The flat face has a tendency to 'walk' across the part surface when plunged. Therefore, to avoid both of these problems, we will use a profile sequence. We will use the same cutting tool from the previous sequence, a 1/8 inch end mill.

Profiling Sequence

Step 1:   Select the Mill "tab" >> Profile Milling from the toolbar.
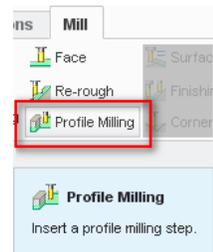
Step 2:   The Profile Milling Dashboard appears.
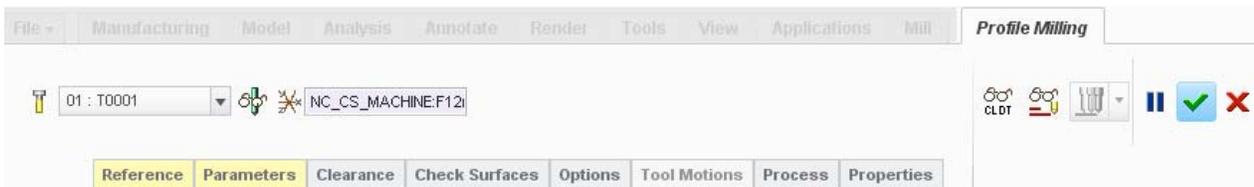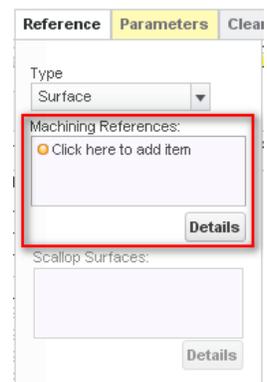
Step 3:   Select T0001 from the Tool Drop Down Menu or select   to create a new tool.

   Note: T0001 should be a 1/8 inch End Mill.
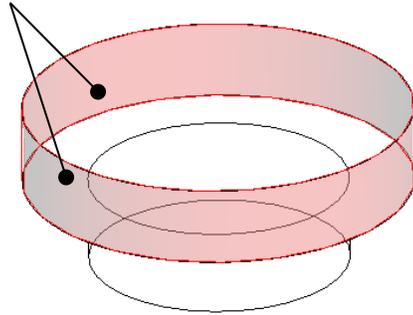
Step 4:   Select the Reference Tab to select the surfaces to machine.

   Note: Type >> set to Surface

Step 5:   To select the Machining Reference surfaces, hold down the Ctrl Key, and select the (4) interior vertical surfaces of the first counterbore hole. Be sure to select both sides of the hole. Then add the surfaces of the second hole. (4 surfaces per hole, 8 total)

First/Second Surface   Third/Fourth Surface

Step 6:   Select the Parameters Tab and select the icon [icon] in the lower right corner of the Parameters Table to Copy machining parameters from another step. Select the Volume Milling from the dialog box.

Parameters	Clearance	Check Surfaces	Opt

CUT_FEED	O
ARC_FEED	-
FREE_FEED	-
RETRACT_FEED	-
PLUNGE_FEED	-
STEP_DEPTH	O
TOLERANCE	0.001
PROF_STOCK_ALLOW	0
CHK_SRF_STOCK_ALLOW	-
WALL_SCALLOP_HGT	0
CUT_TYPE	CLIMB
CLEAR_DIST	O
SPINDLE_SPEED	O
COOLANT_OPTION	OFF

Select Step

1: Volume Milling, Operation: OP020

OK   Cancel

Step 7:   Select ✔ to apply and make changes.

Step 8:   Select LMB (left mouse button) the Profile Milling sequence from the Model Tree and hold down your RMB, Select >> PLAYPATH >> SCREENPLAY and verify that the tool profiles around the edges selected.

Step 9:   Save your manufacturing model.

Profiling Sequence

Within this section you will create an NC sequence to profile the outer surfaces of the part. This is a simple method used to machine the part from the board. Profile milling is used to rough or finish mill vertical or slanted surfaces. The surfaces selected must allow for a continuous tool path. The depth of the cut is defined by the depth of the selected surfaces.

Step 1:   Select the Mill "tab" >> Profile Milling from the toolbar.
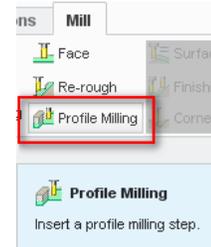
Step 2:   The Profile Milling Dashboard appears.

Step 3:   Select T0001 from the Tool Drop Down Menu or select [tool icon] to create a new tool.

Note: T0001 should be a 1/8 inch End Mill.

Step 4:   Select the Reference Tab to select the surfaces to machine.

Note: Type >> set to Surface

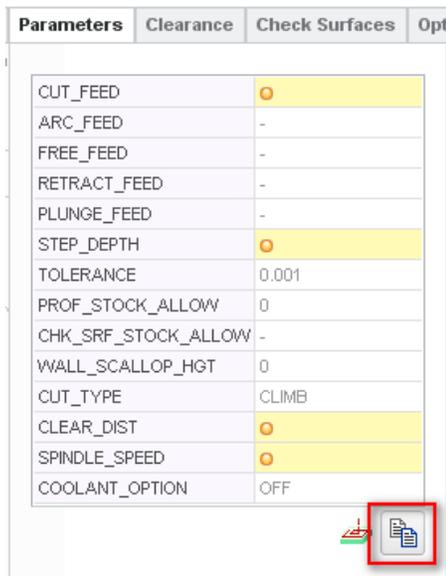**Step 5:**   Select the Machining Reference surfaces, hold down the Ctrl Key, and select the (2) outer vertical surfaces of the part.
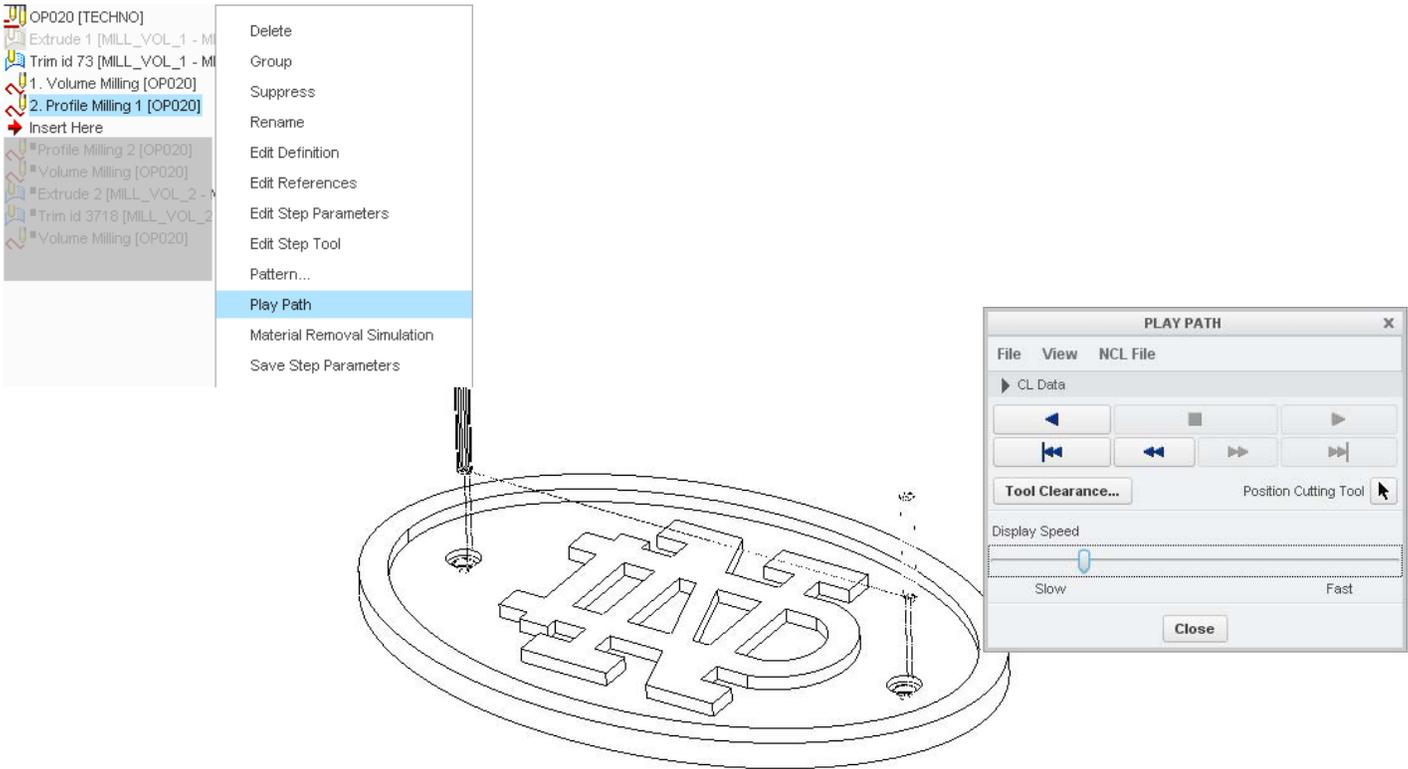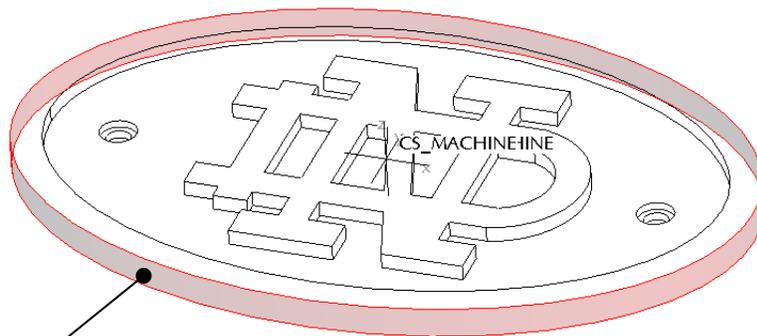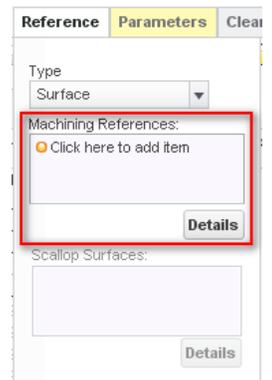
Select Outer Surfaces

Step 6: Select the Parameters Tab and select the icon in the lower right corner of the Parameters Table to Copy machining parameters from another step. Select the Profile Milling 1 from the dialog box.
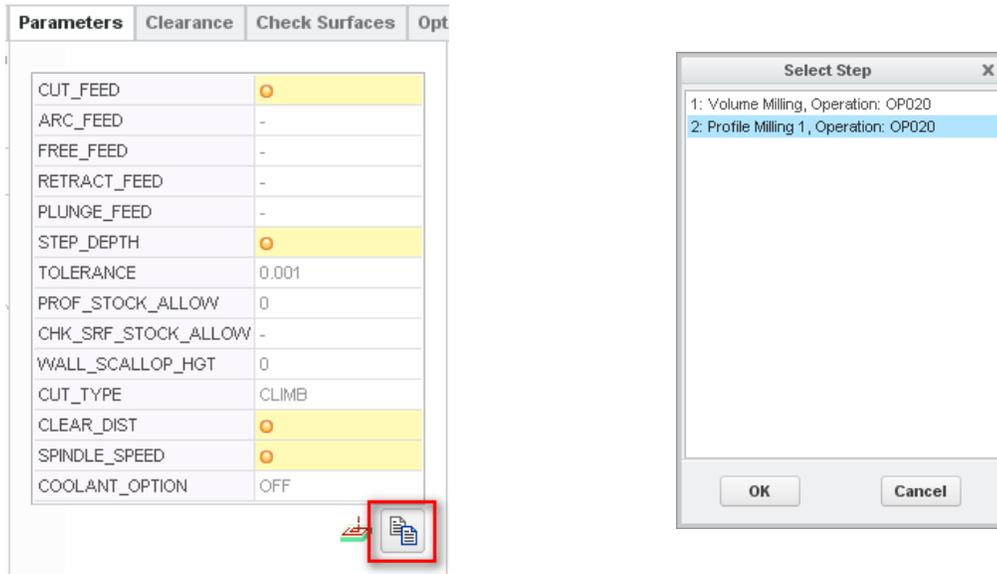
Step 7: Select ✔ to apply and make changes.

Step 8: Select LMB (left mouse button) the Profile Milling 2 sequence from the Model Tree and hold down your RMB, Select >> PLAYPATH >> SCREENPLAY and verify that the tool profiles around the edges selected.
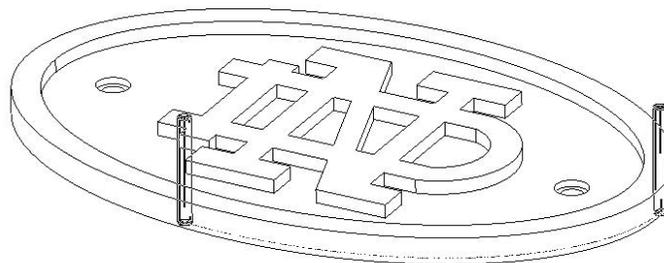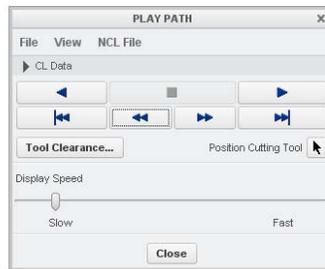

Step 9: Save your manufacturing model.

Outputting the Centerline (CL) Data for Individual Sequences

Within this segment of the tutorial, you output the CL data for each sequence individually, beginning with the Volume Milling sequence.

Step 1:   Select the Manufacturing Tab and select >> Save a CL File

Step 2:   From the Menu Manager select >> NC SEQUENCE

   Note: output Sequences individually … not as an Operation

Step 3:   From the NC SEQ LIST select >> VOLUME MILLING

Step 4:   Select File from the Path menu.

Step 5:   On the Output Type menu, be sure the CL FILE, INTERACTIVE and COMPUTE CL options are selected.

Step 6:   Select Done on the Output Type menu.

Step 7:   In the Save a Copy dialog box, enter Volume as the New Name for the CL file, then select OK.

Step 8:   Select Done Output

   Next, you will post-process the CL data to a specific machine tool.

Step 9:   From the top menu bar (Manufacturing Tab) select >> Post a CL File

Step 10:   Select the file Volume.ncl from the Open Dialog box, select Open to close the window. In the PP Options menu make sure the Verbose and Trace options are selected.

Step 11:   Select Done on the PP OPTIONS menu and a list of post-processors appears.

   Techno Mill (all) use this post: uncx01.p20   (outputs a **.NC file)
   Dyna Mill use this post: uncx01.p26   (outputs a **.TAP file)

Step 12:   Select UNCX01.p20

Step 13:   Select Close to close the information.

Step 14:   Save your manufacturing file.

   The post-processed file created (Volume.nc) is found in your current working directory. This text file may be edited using a standard text editor if any changes are necessary.

Step 15:   Repeat the above process for the other sequences you created … name the files Holes and Profile.

Step 16: Be sure to save the files in a directory which you have access to from webfile, etc.

You should have the following files in your directory:

1. volume.nc
2. holes.nc
3. profile.nc

Adding a Workpiece

In this segment of the tutorial you will learn how to add a workpiece to your model.

To enable visualization of the machining process it is beneficial (though not essential) that the stock material from which this part will be machined, is defined. This is called a "Workpiece". The workpiece represents the raw material or stock that is going to be machined by the manufacturing operations. Its use is optional in NC Manufacturing. NC Manufacturing allows you to create or assemble the workpiece. You can manually create the workpiece (manual workpiece) or allow NC manufacturing to automatically create it using standard shapes and your choice of dimensions (automatic workpiece). Alternatively, you can assemble the workpiece using an existing design model. In this case, the workpiece can have features inherited or merged from the design model.

When you assemble or create a workpiece, or classify a component as a workpiece, it is displayed in a transparent green. This helps visually distinguish between the reference part and the workpiece geometry.

Step 1:   Drag the Insert Here bar in the Model Tree to just below the ND_LOGO part

Step 2:   Select Insert >> Workpiece >> Auto-Workpiece or the icon located at the top of the right-hand toolbar


Step 3:   Select ✔ to apply and make changes.

Next … we need to edit our configuration file to enable visualization of the machining process since adding a workpiece

Step 4:   Select File >> Options >> Configuration Editor (you may need to export/import a copy of this file so you can edit it) >> Add >> (Options Name) nccheck_type >> (Options Value) nccheck, select OK >> OK (apply changes to config file). This option allows you to view a step-by-step simulation of material removal as the tool is cutting the workpiece.

Step 5:   Now drag the Insert Here bar to the bottom in the Model Tree
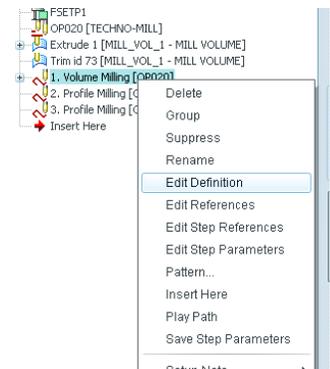
Step 6:   Select the Volume Milling Sequence and (RMB) Edit Definition

Step 7:   Select Play Path >> NC Check >> Step Size >> Enter >> Enter >> .05

Step 8:   Select Run …

Note: The material is removed according to the actual tool shape and motion, that is, all scallops left by the tool passes will be displayed.

This sequence is now complete … SAVE EVERYTHING!