Table of Contents

Auto-Drilling Tutorial

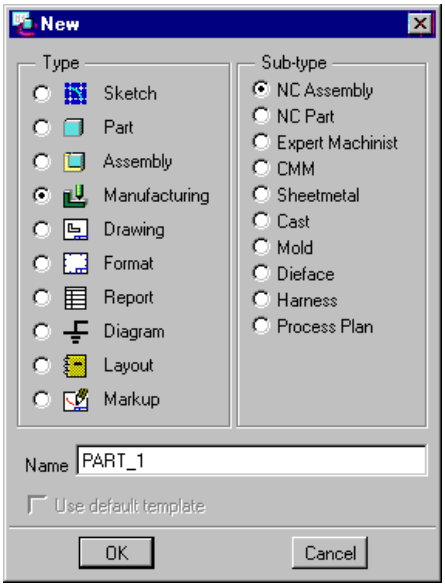
Introduction

Auto Drilling is intended to ease the creation of hole-making sequences and to reduce the time required to program the tool paths. This will be accomplished through the use of a new interface which will allow the programmer to create multiple NC hole-making sequences from one dialog box with a minimum of picks. Auto Drilling will automatically create the NC hole-making sequences, on holes chosen by the user, by referencing the information in an appropriately assigned Manufacturing UDF. This will all take place from within one table/dialog box. Once created, the NC sequences can be edited manually through standard Pro/NC methods.

Creating NC Sequence

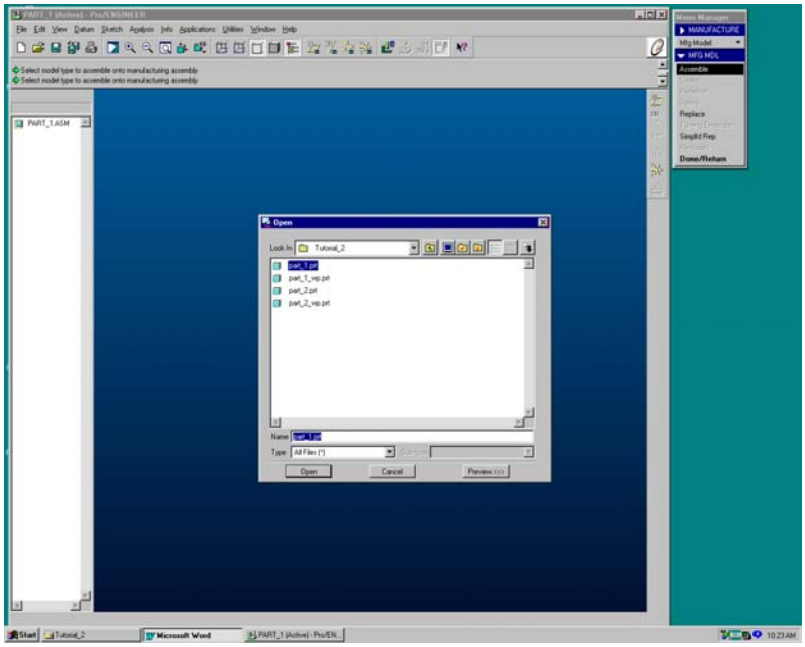
All of the files for this tutorial are attached.

Retrieve the file, *part_1.prt, part_2.prt, part_1_wp.prt, part_2_wp.prt* and save them in a new folder. Then go to **New > Manufacturing > NC Assembly**, give the file name *PART_1***.** Click **OK.**



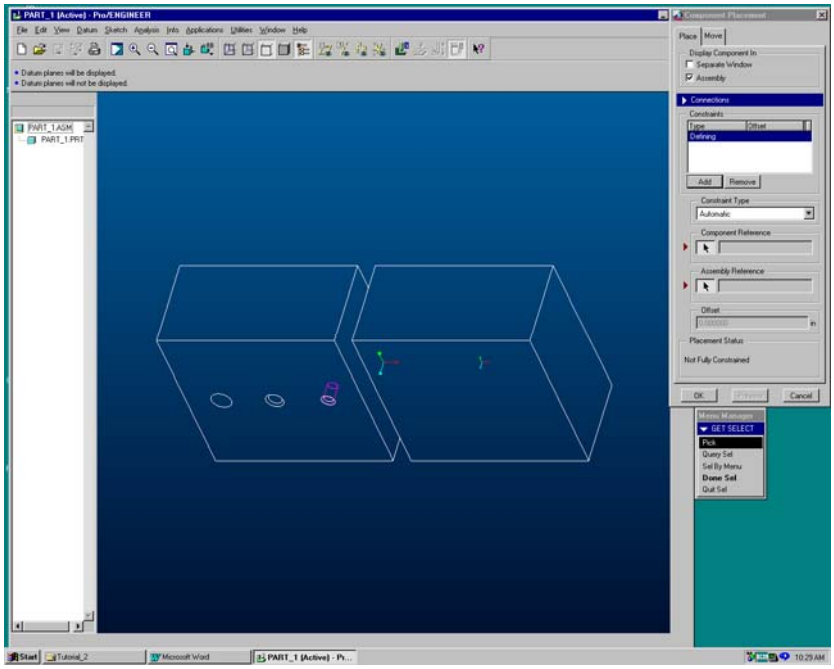
- Step 1. Assembly *part_1 and part_1_wp* (part and its workpiece)

MANUFACTURE > Mfg Model > Assemble > Ref Model



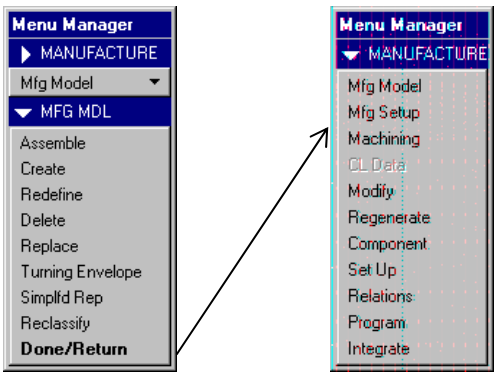
Select *part_1.prt*, then click **Open**.

Assemble > Workpiece, select *part_1_wp.prt*, then click **Open**



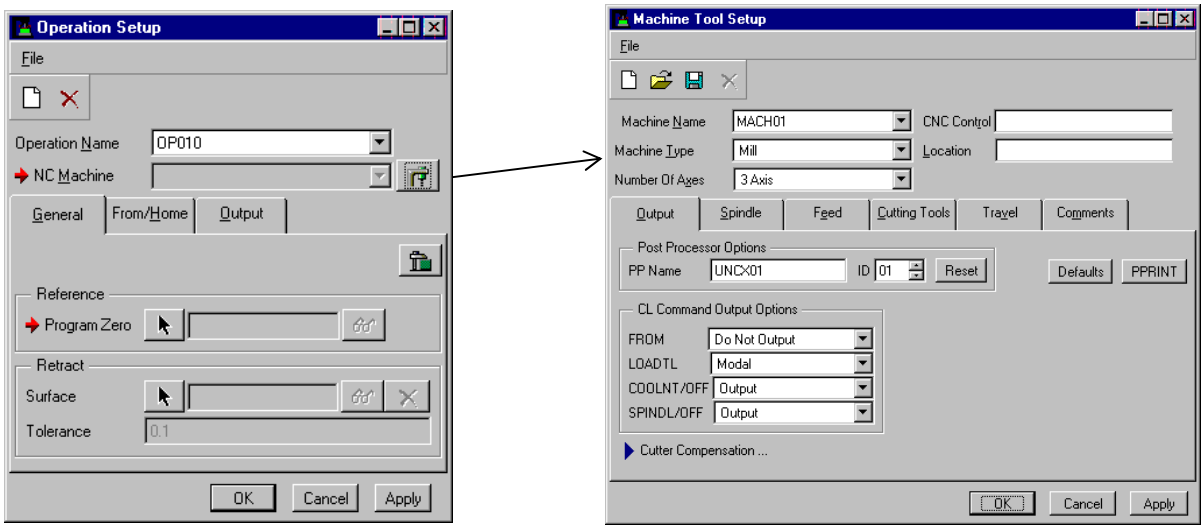
Assemble part_1 and part_1_wp. (See Pro-e ‘assembly tutorial’ for detail)

- Step 2. Manufacturing Setup



After assembly, click **Done/Return**, then **Mfg Setup**

For choosing machine, click the icon right to NC Machine

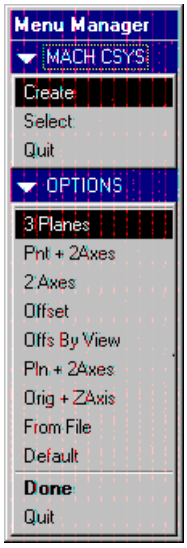


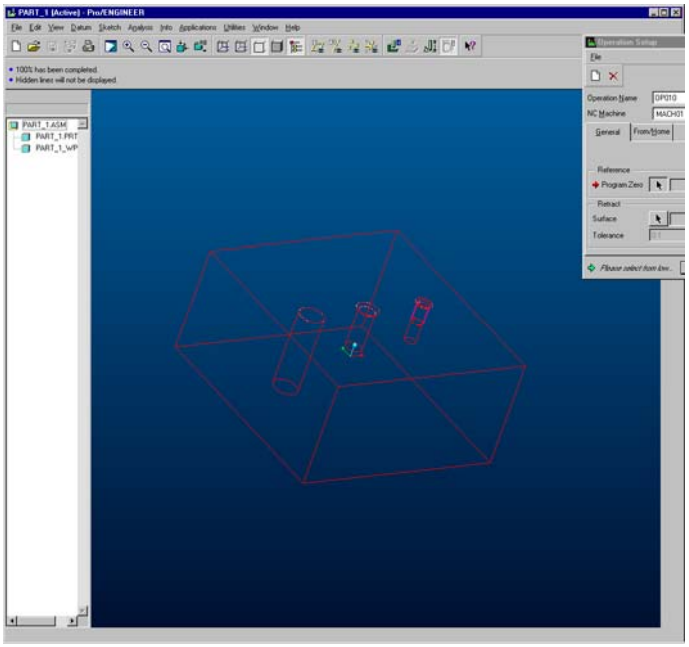
Select **Mill, 3 Axis**, and keep other contents as defaults, then click **OK**

For creating programming zero, click icon right to Program Zero

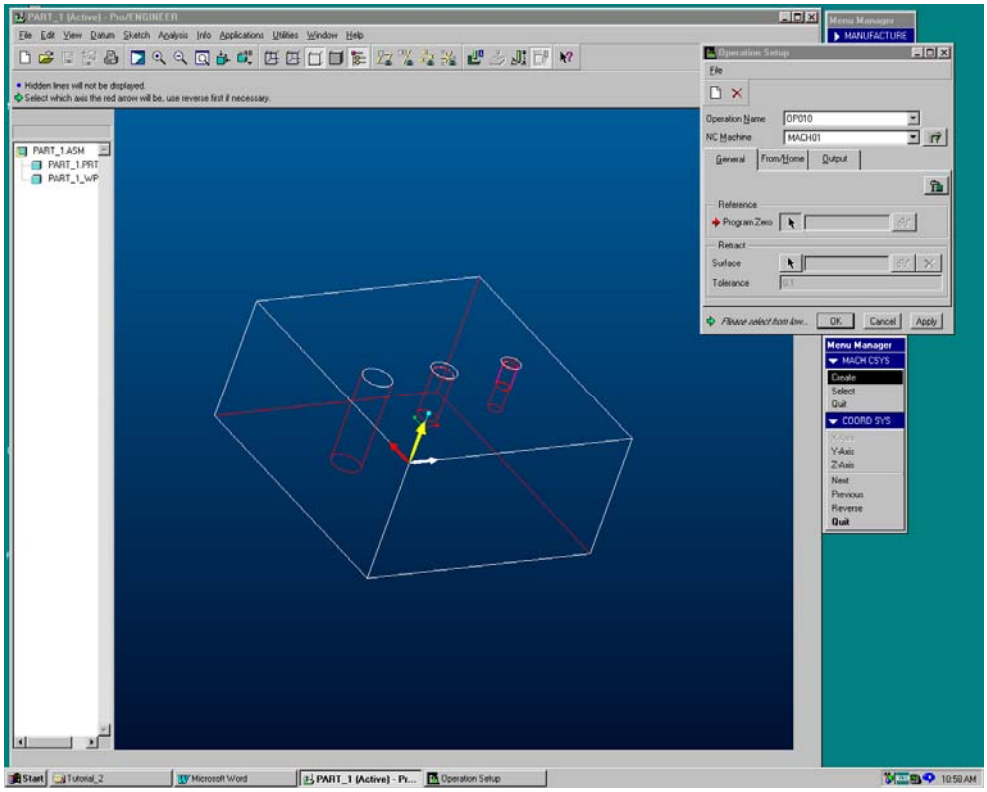
Creat > Pick, then pick workpiece

3Planes > Done > Plane > Pick



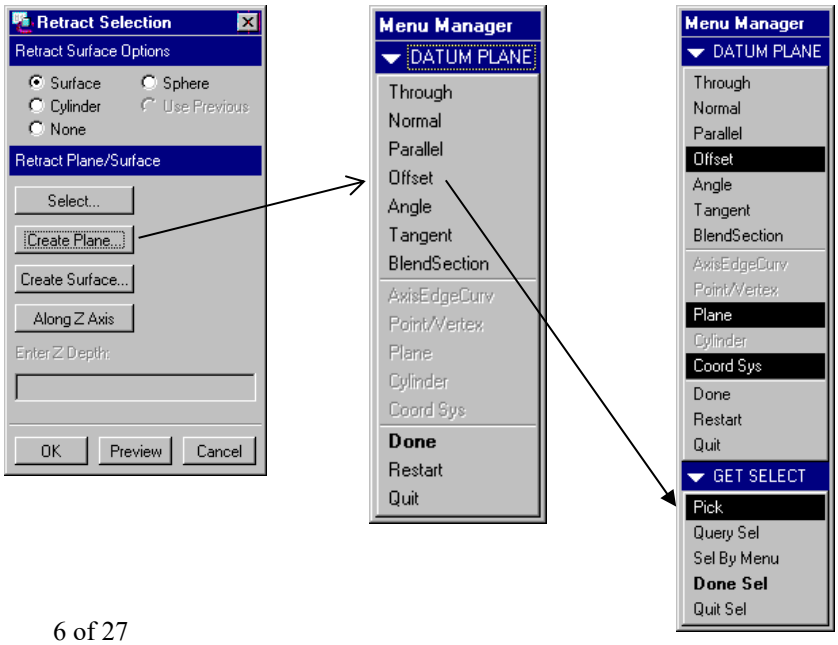


Reorient your assembly as the above window shows, then pick left, front and top plane

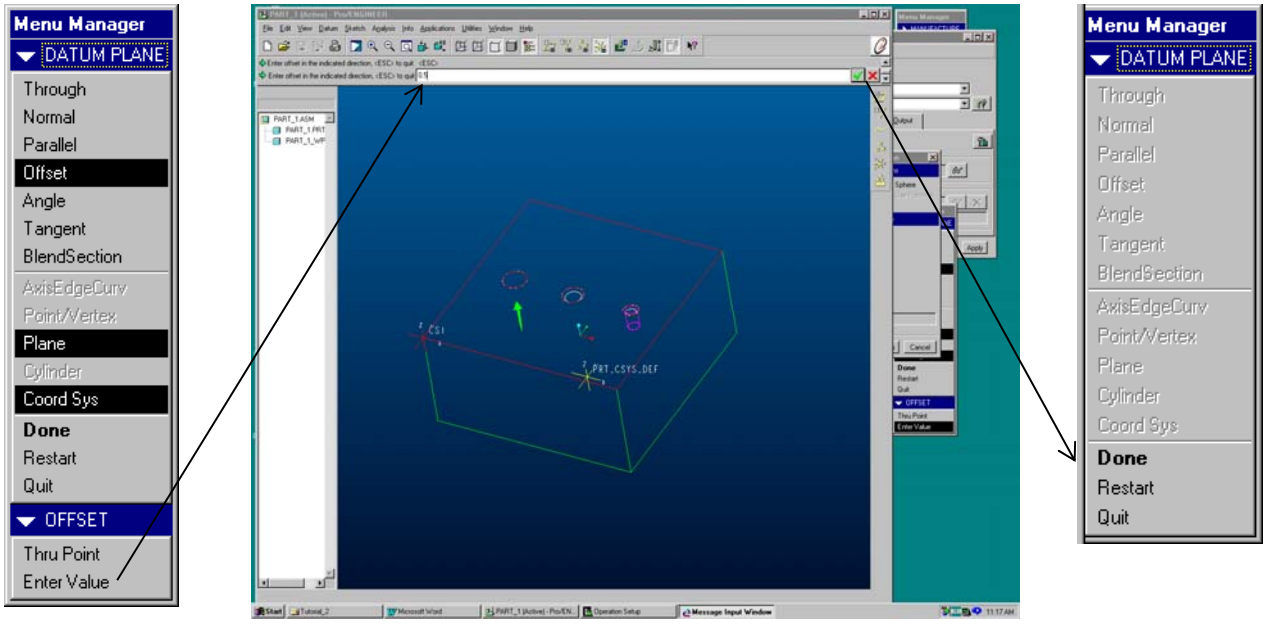


Using **Next and Reverse** to choose and reorient X,Y, Z axis, Let X pointing right and Y pointing back just same as the above figure shows. The selected axis is in color red, when it is oriented correctly, click the corresponding axis. CS1 is created.

For creating retract plane, click icon right to Retract surface, then Retract Surface Options box comes. Click **Create Plane > Offset > Pick**



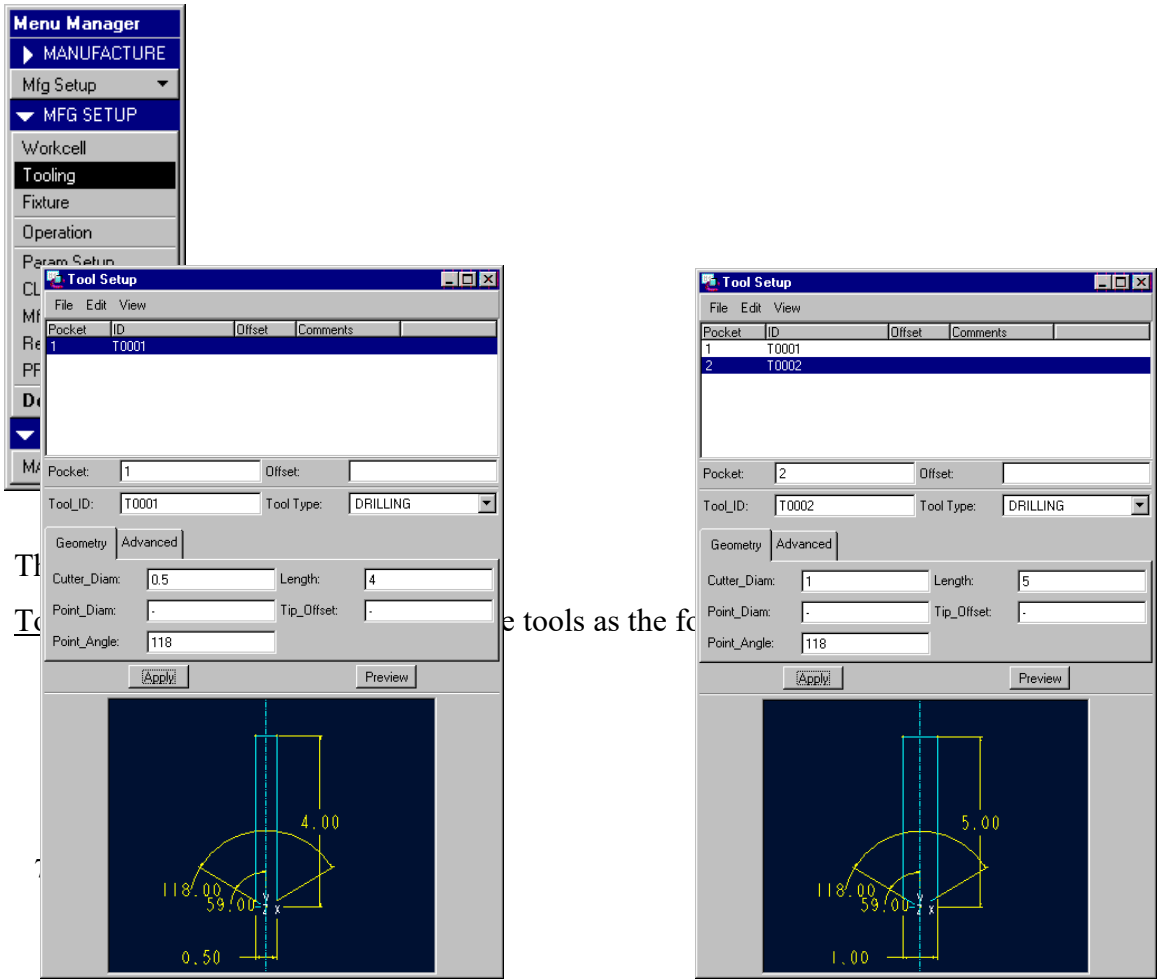
Pick the top surface of the work-piece

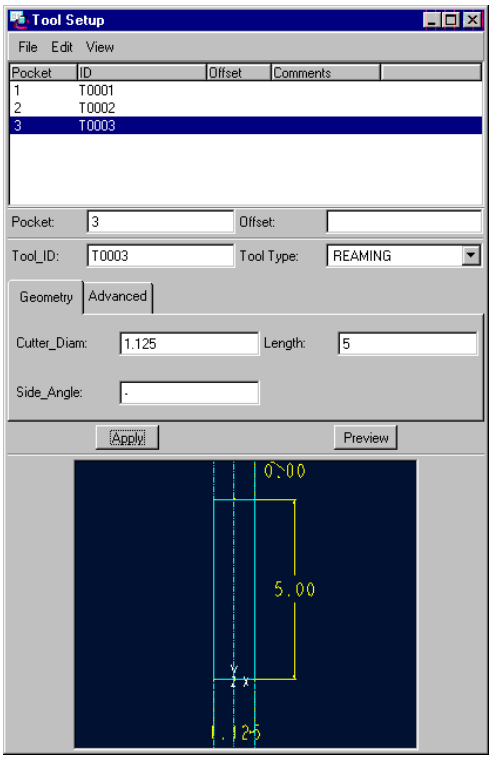


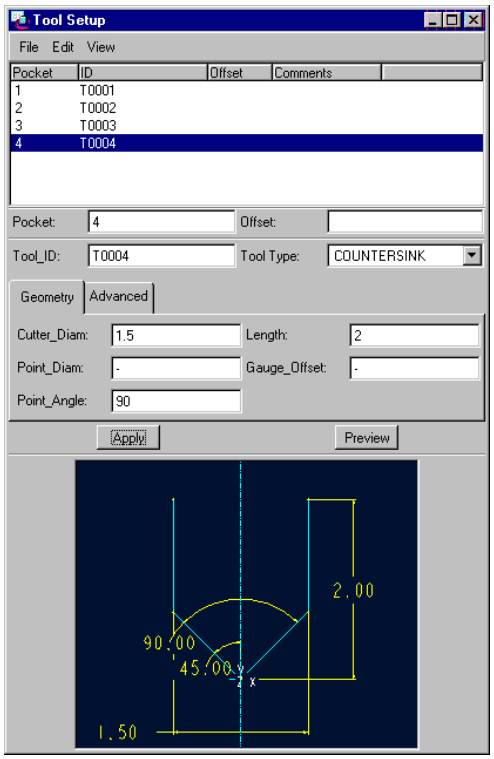
Then click **Enter Value**, put 0.5 then **Enter,** then **Done**

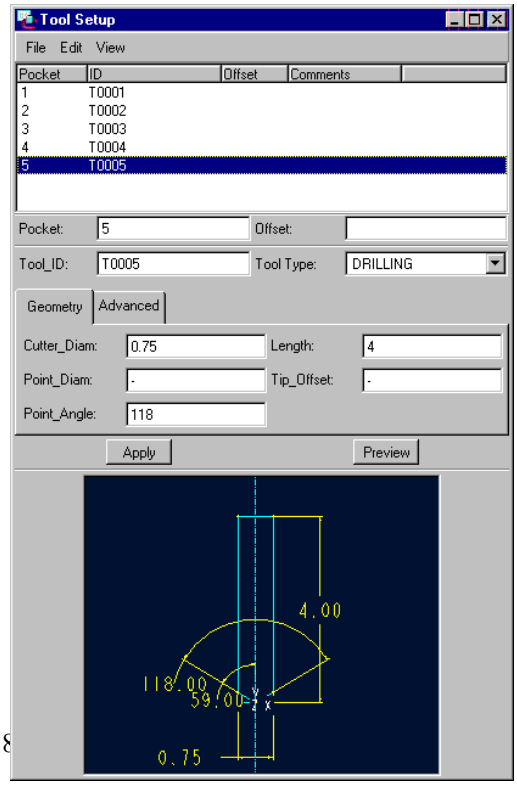
Click **OK** in the Retract Surface Options box

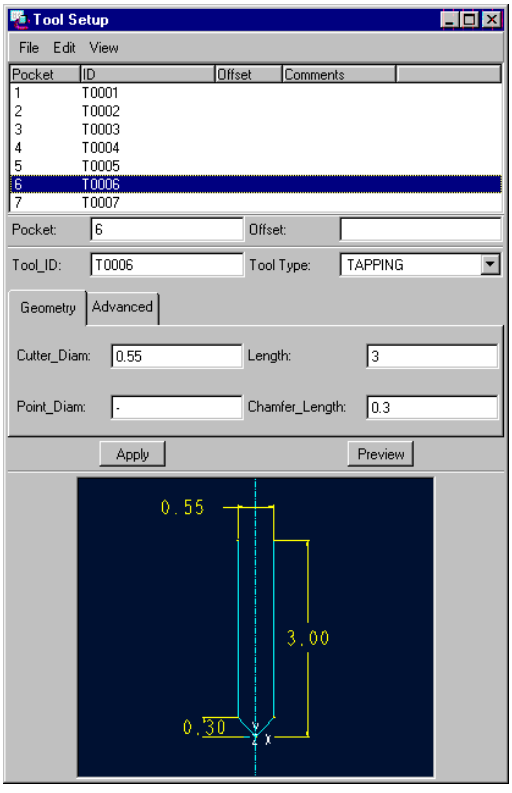
Put 0.01 for the tolerance, then click **OK** in the Operation Set Box

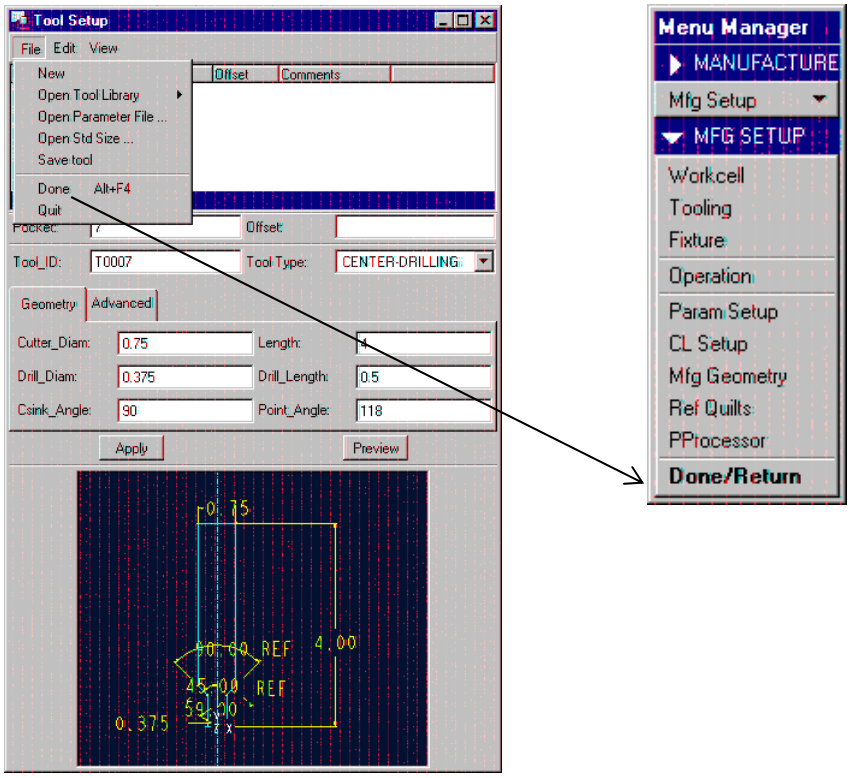



Tool Setup
File Edit View
Pocket ID Offset Comments
1 T0001
2 T0002
3 T0003
Pocket: 3
Offset:
Tool_ID: T0003
Tool Type: REAMING
Geometry Advanced
Cutter_Diam: 1.125
Length: 5
Side_Angle: -
Apply
Preview
0.00
5.00
1.125


Tool Setup
File Edit View
Pocket ID Offset Comments
1 T0001
2 T0002
3 T0003
4 T0004
Pocket: 4
Offset:
Tool_ID: T0004
Tool Type: COUNTERSINK
Geometry Advanced
Cutter_Diam: 1.5
Length: 2
Point_Diam: -
Gauge_Offset: -
Point_Angle: 90
Apply
Preview
2.00
90.00
45.00
1.50


Tool Setup
File Edit View
Pocket ID Offset Comments
1 T0001
2 T0002
3 T0003
4 T0004
5 T0005
Pocket: 5
Offset:
Tool_ID: T0005
Tool Type: DRILLING
Geometry Advanced
Cutter_Diam: 0.75
Length: 4
Point_Diam: -
Tip_Offset: -
Point_Angle: 118
Apply
Preview
4.00
118.00
59.00
0.75


Tool Setup
File Edit View
Pocket ID Offset Comments
1 T0001
2 T0002
3 T0003
4 T0004
5 T0005
6 T0006
7 T0007
Pocket: 6
Offset:
Tool_ID: T0006
Tool Type: TAPPING
Geometry Advanced
Cutter_Diam: 0.55
Length: 3
Point_Diam: -
Chamfer_Length: 0.3
Apply
Preview
0.55
3.00
0.30

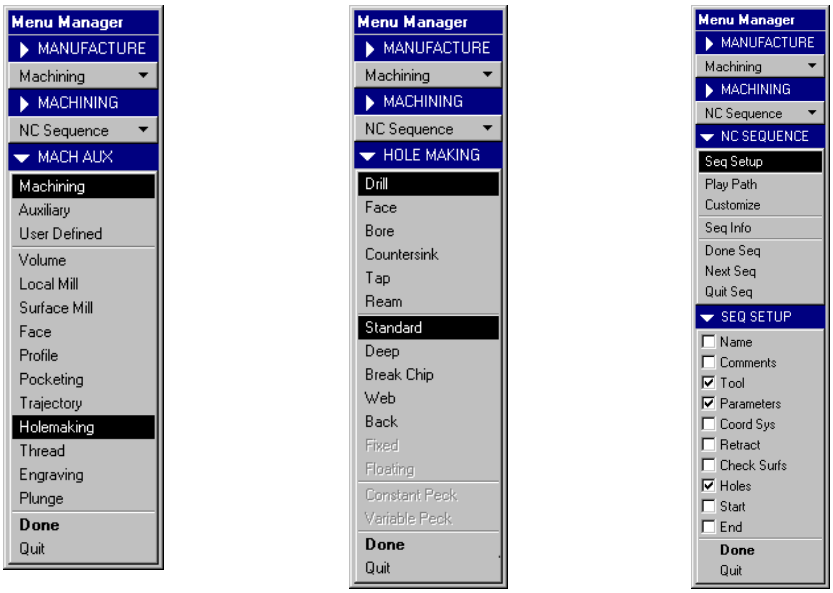


After setting up the 7 tools as the above windows shown, click **Done** from File menu **Done/return** from MFG SETUP box

- Step 3. Creating NC Sequences

Sequence #1

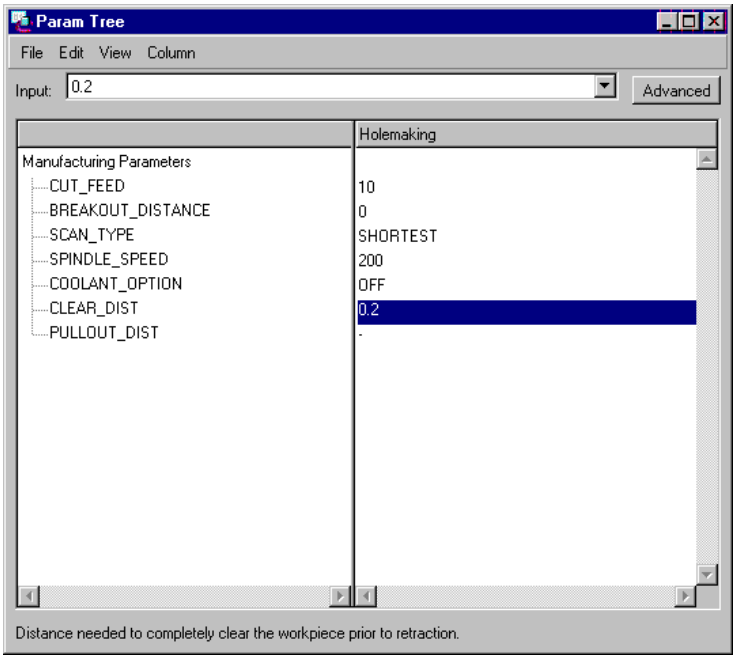
MANUFACTURE > Machining > NC Sequence > Hokemaking > Done



Then **Drill > Standard > Done**

Check Tool, Parameters and Holes, then **Done,** Tool Setup box comes now.

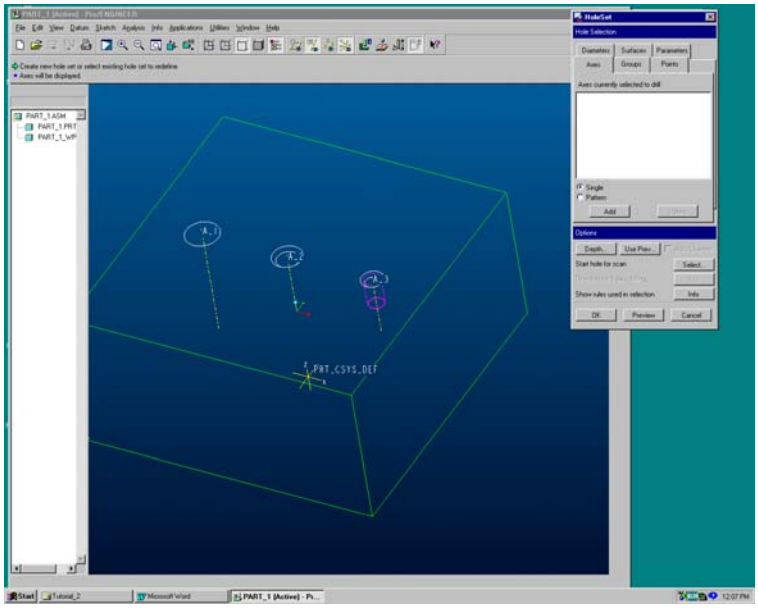
Select Tool #7, center-drill, then **Done** from File Menu



Click **Set,**

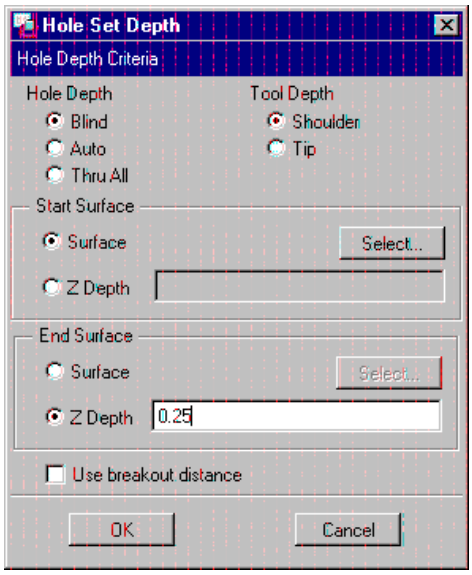
Then set up the parameters as above, click **Exit** from File menu

Click **Done** from MFG PARAMS menu, HoleSet window comes now.



Click **Add,** then pick Axis A1 (most left one) then **Done Sel**, (using Query Sel, if necessary)

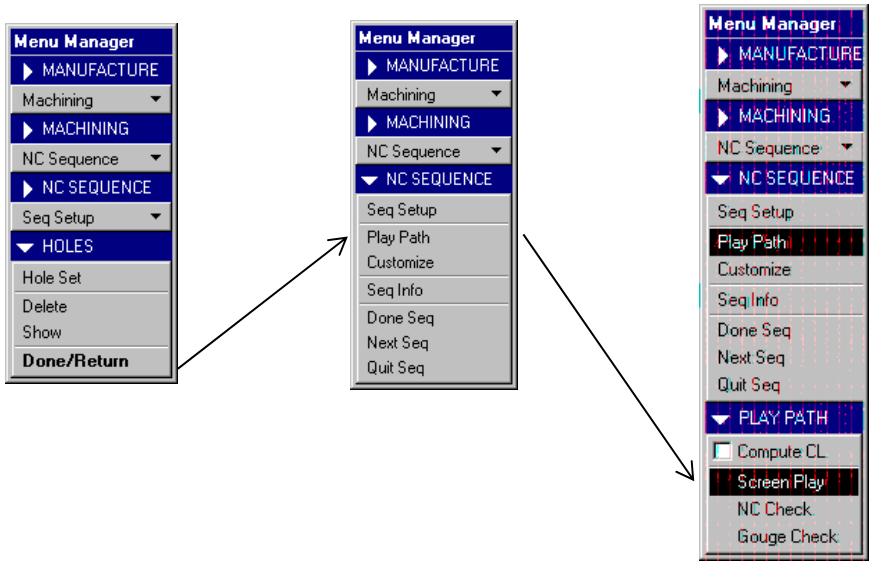
Click **Depth,** check Blind for Hole Depth



Check Surface for Start Surface, click **Select**, click top surface of the workpiece

Check Z Depth for End surface, put 0.25

Click **OK**, in the Hole Set Depth box, click **OK** in the HoleSet box.



Click **Done/Return > Play Path > Screen Play**, then click '**play**' button in the PLAY PATH box, adjust play speed if necessary. Close the PLAY PATH box. Click **Done Seq** in NC SEQRENCE menu

Sequence #2:

Then click **NC Sequence > New Sequence > Holemaking > Done**

Drill > Standard > Done

Check Parameter and Holes only, Then **Done**

Click **Use Prev,** click Sequence #1 in NC SEQ LIST, then **Done**

Then **Axes > Add**, select Axis A2 and A3, **Done Sel**

Click **Depth,** check Blind, for Start and End Surface, same as previous sequence. Then **OK** in the Hole Set Depth box, **OK** in the Hole SetBox

Done/Return > Play Path , close the Play Path box, then click **Done Seq.**

Sequence #3:
NC Sequence > New Sequence > Hole Making > Done

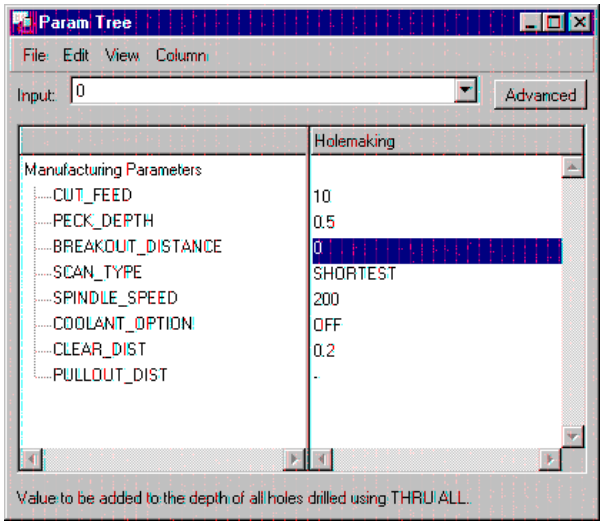
Drill > Deep > Constant Peck > Done

Check Tool, Parameters and Holes, and then **Done**

Select Tool #1 (1/2 inch drill) and the **Done** from File menu

Use Prev, then click sequence 2

Click **Set** in MFG PARAMS menu



Then change PECK_DEPTH to 0.5 and keep other values, click **Exit** from File menu

Then **Done** from MFG PARAMS menu. HoleSet box comes now.

Click **Axes > Add**, select A1 axis, **Done Sel**, then click **OK** in the Hole Set box

Then **Done/Return** in the HOLES menu

You can play the tool path now, then **Done Seq** in NC SEQUENCE menu

Sequence #4:

NC Sequence > New Sequence > Holemaking > Done

Drill > Standard > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #2 (1 inch drill), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #3 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Axes > Add in Hole Selection box, select axis A1, then **Done Sel**

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #5:

NC Sequence > New Sequence > Holemaking > Done

Ream > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #3 (1.125 inch ream), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #4 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Axes > Add in Hole Selection box, select axis A1, then **Done Sel**

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #6:

NC Sequence > New Sequence > Holemaking > Done

Drill > Standard > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #5 (0.75 inch drill), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #3 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Axes > Add in Hole Selection box, select axis A2, then **Done Sel**

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #7

NC Sequence > New Sequence > Holemaking > Done

Countersink > Done

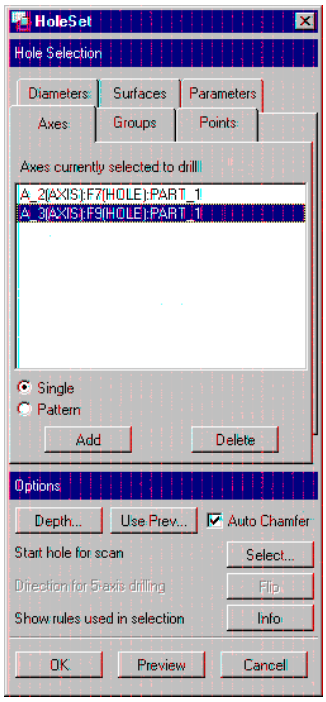
Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

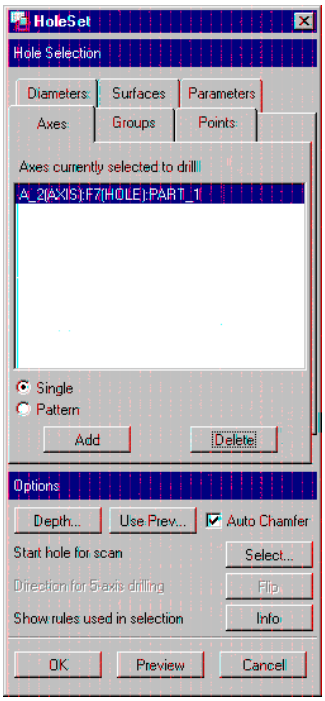
Select Tool #4 (countersink), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #6 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Check Auto Chamfer, then axis A2 and A3 are automated selected, because they have chamfer feature. Click on A3 and then **Delete,** keep A2 only





OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #8:

NC Sequence > New Sequence > Holemaking > Done

Drill > Standard > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #1 (0.5 inch drill), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #3 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Axes > Add in Hole Selection box, select axis A3, then **Done Sel**

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #9:

NC Sequence > New Sequence > Holemaking > Done

Countersink > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #4 (countersink), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #7 under NC SEQ LIST

Then click **Done** in MFG PARAMS menu,

Check Auto Chamfer, in Hole Selection box, then axis A2 and A3 are automated selected, because they have chamfer feature. Click on A2 and then **Delete,** keep A3 only

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu

Sequence #10:

NC Sequence > New Sequence > Holemaking > Done

Tap > Fixed > Done

Then check Tool, Parameters and Holes in the SEQ SETUP menu, click **Done**

Select Tool #6 (Tapping), then **Done** from File menu

Use Prev in MFG PARAMS menu, click NC sequence #9 under NC SEQ LIST

Click **Set** in MFG PARAMS menu, change THREAD FEED to 5, and keep others

Then **Exit** from File menu

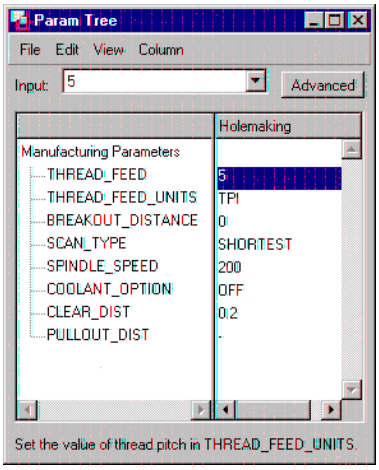
Then click **Done** in MFG PARAMS menu

Axes > Add in Hole Selection box, select axis A3, then **Done Sel**

OK in Hole Selection box

Done/Return in HOLES menu, then you can play the tool path

Done Seq in NC SEQUENCE menu



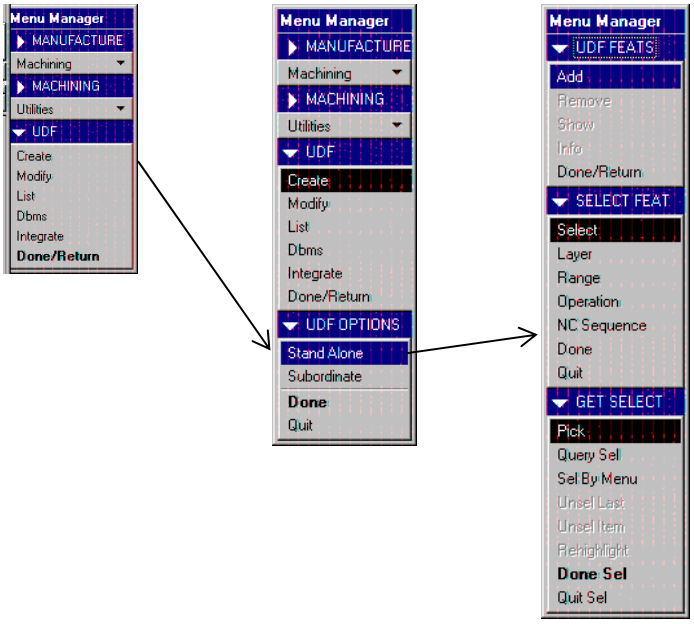
We finished all of the NC sequence, it is ready to create MUDF (manufacturing user defined feature)

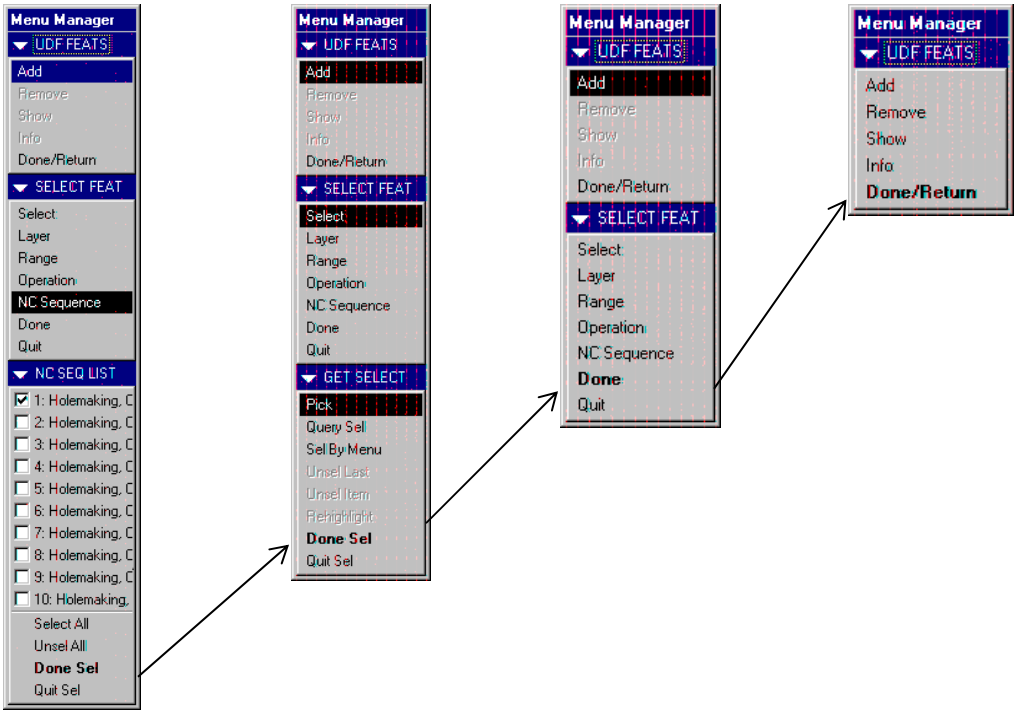
Creating MUDF

- MUDF #1:

Machining > Utilities > UDF Library > Create, give MUDF name center-drill, then enter

Stand Along > Done





Click **NC Sequence**, check NC sequence #1, **Done Sel**

Click **Done Sel** in NC SEQ LIST menu

Click **Done Sel** in GET SELECT menu

Click **Done** in SELECT FEAT menu

Click **Done/Return** in UDF FEATS menu

Click **Single** in Prompts menu, then **Done/Return**

For prompt, enter

'operation' for [operation]

'oper_csys' for [coordinate system]

'oper_retract' for [retract surface]

'axis' for [axis]

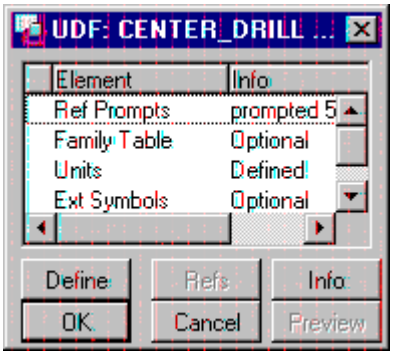
'start_surface' for [start surface]

(Check Single and click 'enter' after each prompt)

(Very important notice: the words entered for prompt must be exact same as the aobve)

Click **Done/Return** in SET PROMPT menu

Click **OK** in UDF setting table



- MUDF #2:

In UDF menu click **Create,** give MUDF name drill_1, then enter

Stand Along > Done

Click **NC Sequence**, check NC sequence #3, #4, #5, **Done Sel**

Click **Done Sel** in NC SEQ LIST menu

Click **Done Sel** in GET SELECT menu

Click **Done** in SELECT FEAT menu

Click **Done/Return** in UDF FEATS menu

Click **Single** in Prompts menu, then **Done/Return**

For prompt, enter same words as MUDF#1

Done/Return in SET PROMPT menu

Click **OK** in UDF setting table

- MUDF #3:

In UDF menu click **Create,** give MUDF name drill_2, then enter

Stand Along > Done

Click **NC Sequence**, check NC sequence #6, #7, **Done Sel**

Click **Done Sel** in NC SEQ LIST menu

Click **Done Sel** in GET SELECT menu

Click **Done** in SELECT FEAT menu

Click **Done/Return** in UDF FEATS menu

Click **Single** in Prompts menu, then **Done/Return**

For prompt, enter same words as MUDF#1

Done/Return in SET PROMPT menu

Click **OK** in UDF setting table

- MUDF #4:

In UDF menu click **Create,** give MUDF name drill_3, then enter

Stand Along > Done

Click **NC Sequence**, check NC sequence #8, #9, #10, **Done Sel**

Click **Done Sel** in NC SEQ LIST menu

Click **Done Sel** in GET SELECT menu

Click **Done** in SELECT FEAT menu

Click **Done/Return** in UDF FEATS menu

Click **Single** in Prompts menu, then **Done/Return**

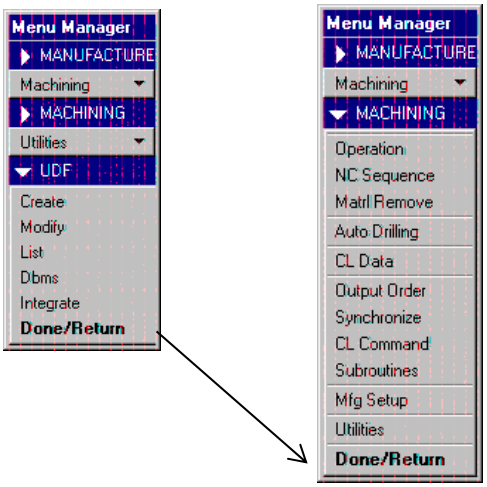
For prompt, enter same words as MUDF#1

Done/Return in SET PROMPT menu

Click **OK** in UDF setting table

Click **Done/Return** in UDF menu

Click **Done/Return** in MACHINING menu



Now four MUDF are done for future use.

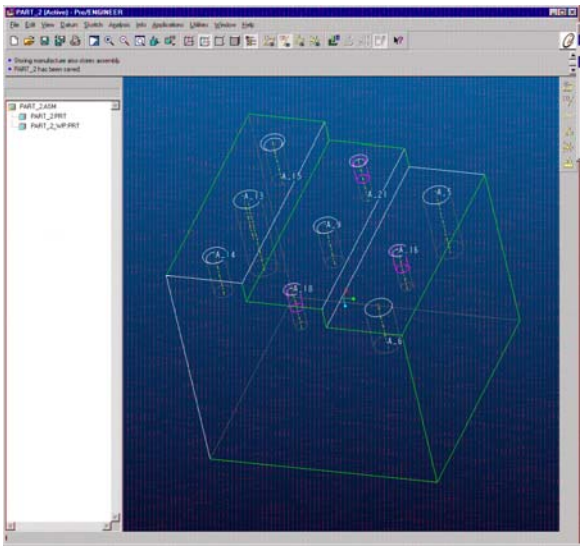
Save your file and close window.

Auto-Drilling

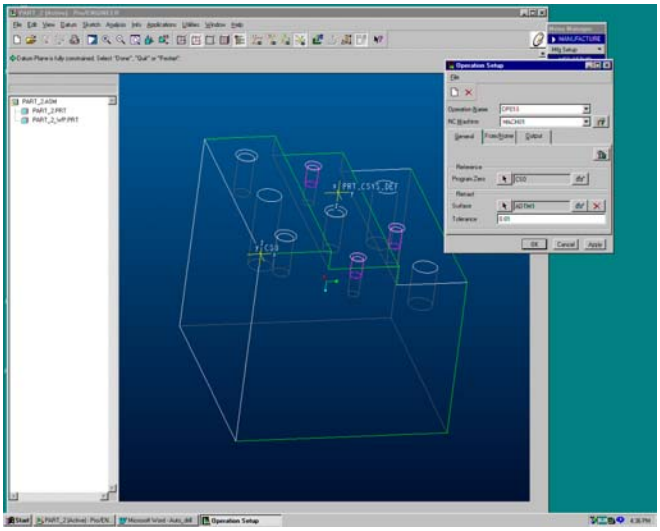
Step 1. Manufacturing Setup:

Click **New**, check Manufacturing > NC Assembly, give file name: part_2, then **OK**

Retrieve part_2.prt and part_2_wp.prt and assembly them.



Click **Mfg Setup,** then operation setup window comes, put OP_PART2 for operation



name, 3 axis mill for machine, CS0 for Program Zero (shown on above picture), retract surface is ½ inch above the most top plane, 0.01 for tolerance. (refer Manufacturing Setup of part_1)

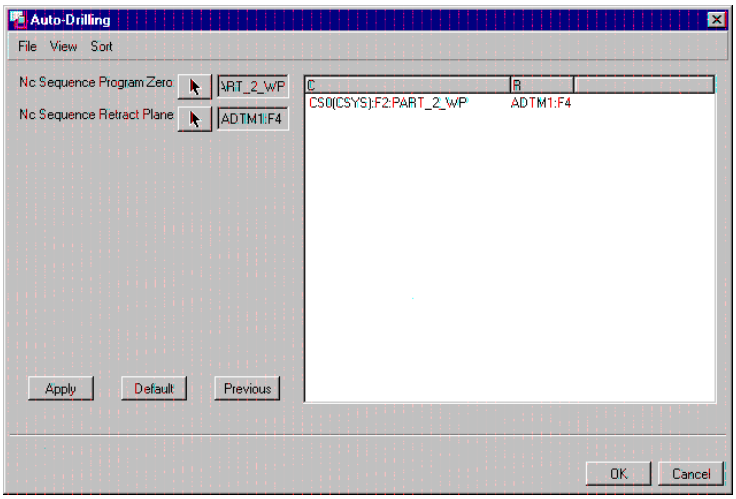
Then click **OK** in Operation Setup table

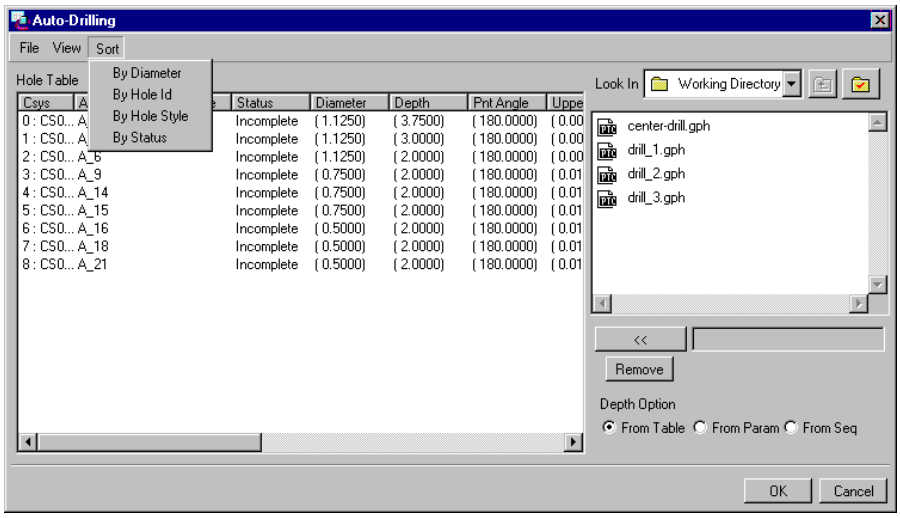
Then **Done/Return** in MFG SETUP menu

Step 2. Auto-Drilling

Machining > Auto Drilling

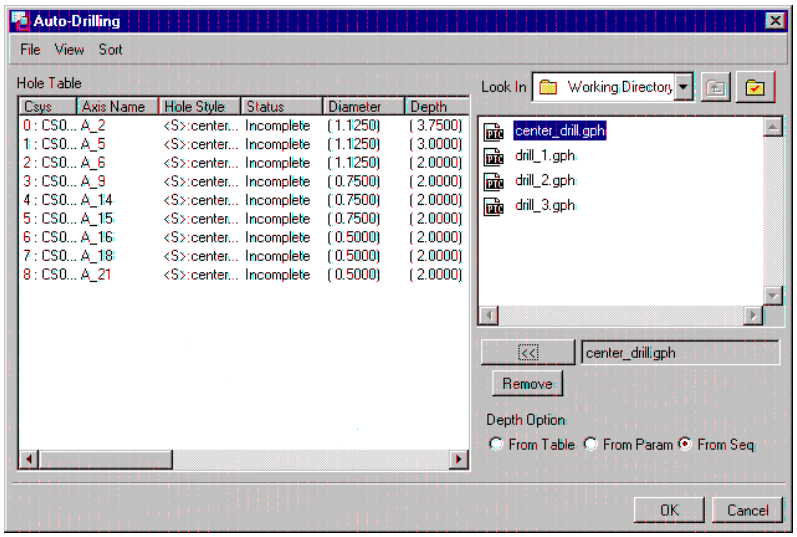
The following window comes, click **OK** to accept the default Program Zero and Retract Plane.





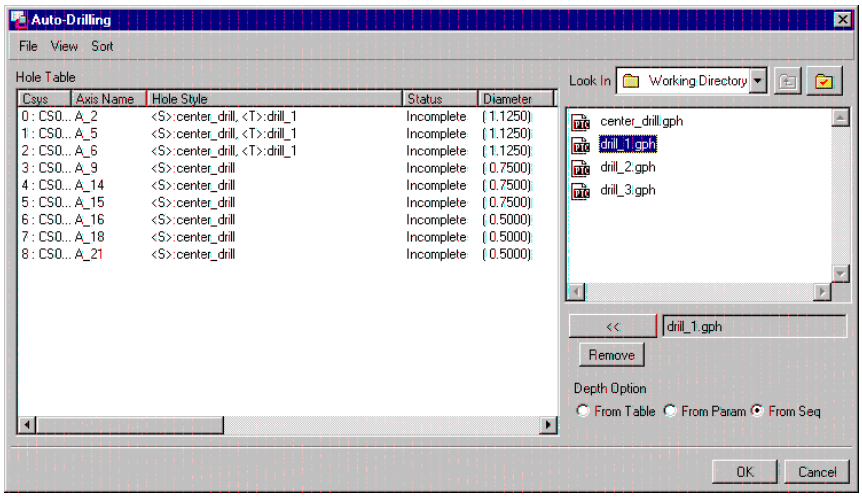
Then Auto-Drilling table comes, you can arrange the rows by clicking **Sort**, (then click by Diameter/Hole Id/Hole Style/Status), in this case it is arranged by Diameters.

Check From Seq for depth option, select **all rows** (They are highlighted after being selected). On right side of the table, there is a MUDF window, select **center_drill.gph**, then click << to apply drill strategy to these rows.

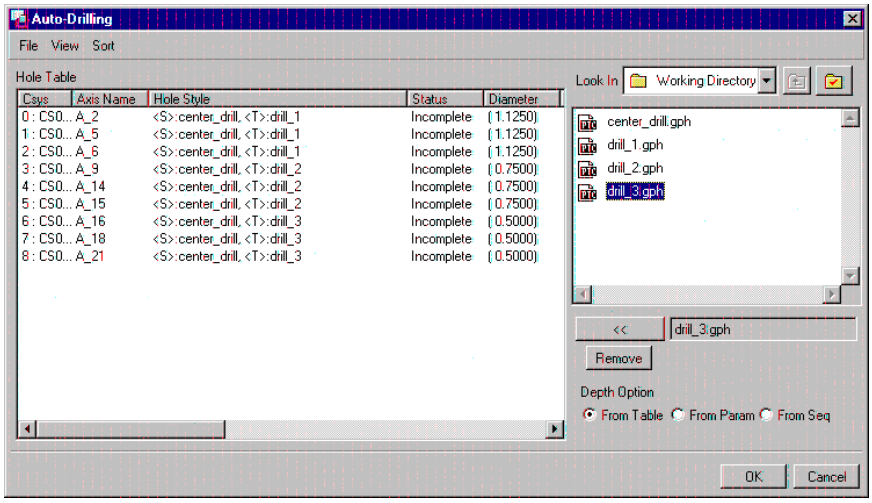


If you make a mistake, you can select rows and then click **Remove** to remove the drilling strategy.

Now Check From Table for Depth Option, select first 3 rows (row #0, #1, #2), then click on drill_1.gph, click << to apply drill strategy to first 3 rows.

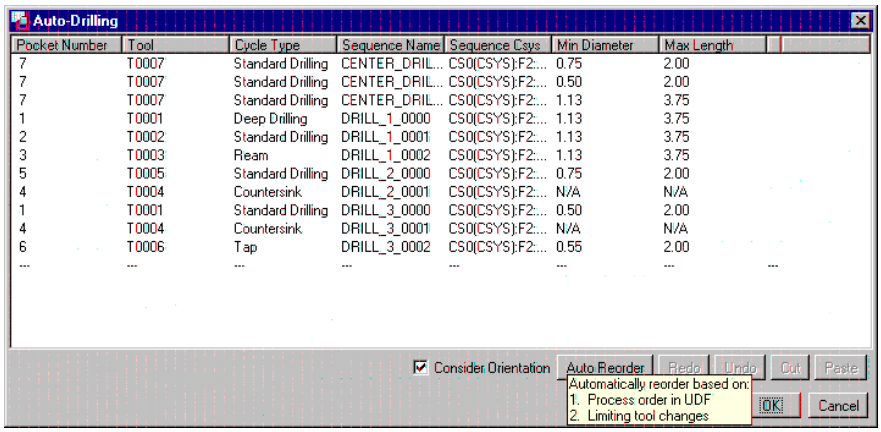


Using the same method to apply strategy drill_2.gph to rows #3, #4, #5, apply strategy drill_3.gph to rows #6, #7, #8.



Now click OK in the Auto Drill table.

Then click **Auto Reorder** to limit tool change.



Click **OK**

Now all the tool path have been created by using Auto-Drilling.

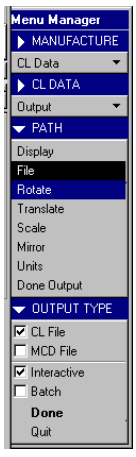
You can go to NC Sequence to check the sequence created.

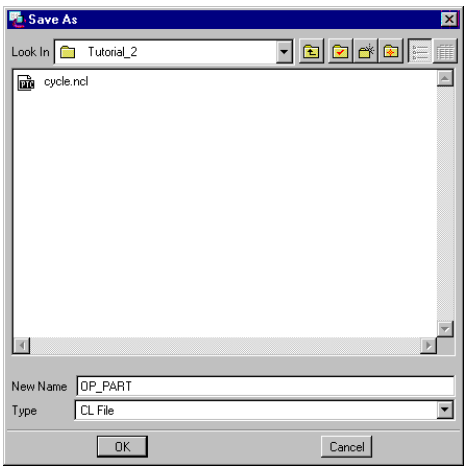
Step 3: Play tool path

CL Data > Output > Select One > Operation > OP_PART > Done

Step 4. Create CL file and simulate with Verycut

CL DATA > Output > Select One > Operation > OP_PART >File





Check CL file only, then **Done**

Give CL file name OP_PART, then click **OK**

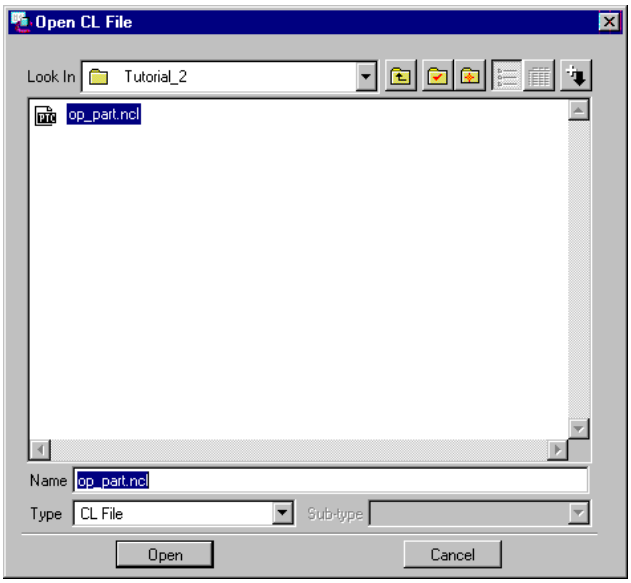
Then **Done Output** from PATH menu

Done/Return from CL Data menu

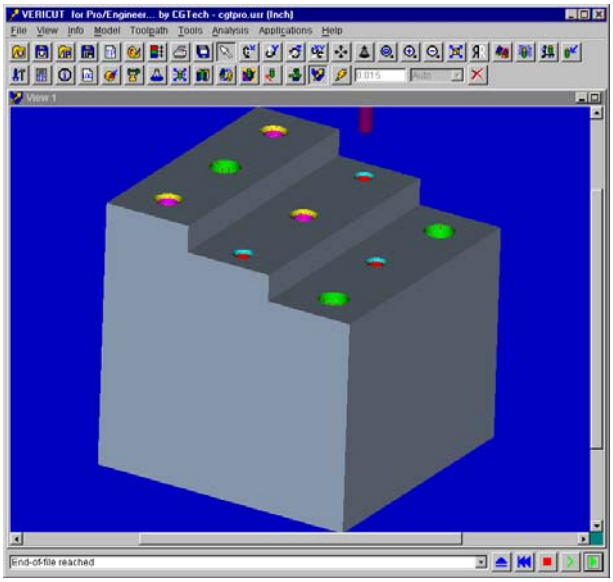
Simulate tool path with Vericut:

CL Data > NC Check > CL File

Choose file op_part.ncl, then Open



Then **Done** in NC VERIFICATION menu



5. Automatically selecting MUDF by setting feature parameters in Proe/Part

By setting Feature Parameter 'holestyle' to the MUDF name without extension, the MUDF will be automatically assigned the hole.

Step 1. Create a your part in Pro/E, part must has a hole feature, give name 'autodrill_1'

Step 2. Set Feature Parameter

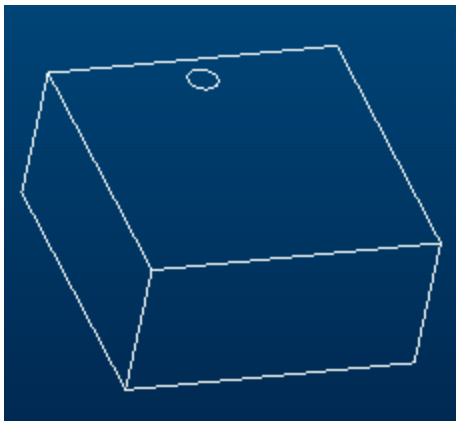
In Part menu, choose **Feature/Setup/Parameters/Feature**

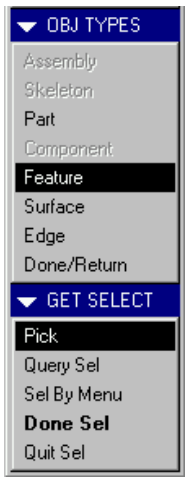
Pick the hole feature, in MODEL PARAMS menu, choose **Create/String**

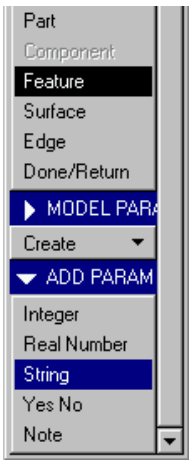
'Enter Parameter Name', give: **holestyle**, then Enter

'Value for HoleStyle', gave: the MUDF name (without extension) you want to use to machine this feature, in this case, give: **drill_1**

Then in Part Setup menu, choose: **Done**







Step 3. Group and pattern the feature

(Using Pattern only, the feature parameters will not be passed to the patterned feature)

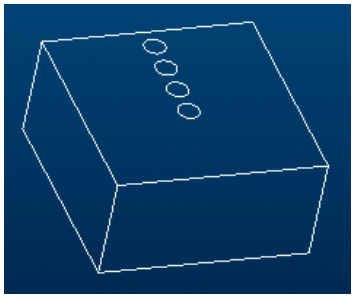
In Part Menu, choose, **Feature/Group/Create/Local Group**

Give a Group Name, then pick the hole (feature)

Then in Select Feat menu choose: **Done Select/Done**

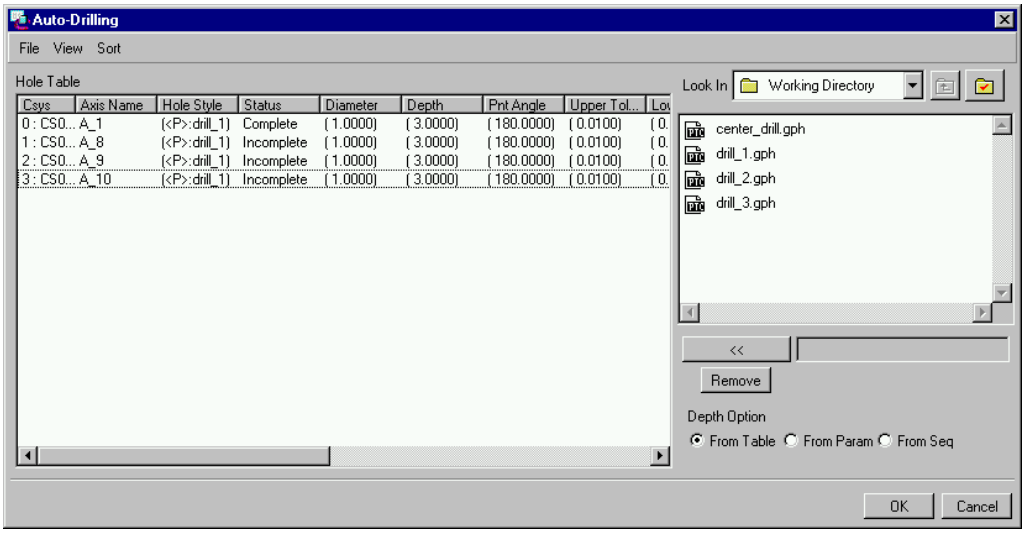
In Group Menu, choose: **Done/Reture**

Pattern the group feature to totally 4 holes



Step 4. In Pro/NC, create a manufacturing file using part: 'autodrill_1.prt' you just created.

In Pro/NC, Manufacture menu choose: **Machining/AutoDrilling**, then in AutoDrilling window, you will see MUDF 'drill_1' is already assigned to each hole, click **OK** in this window.



In the new window, click**: OK**