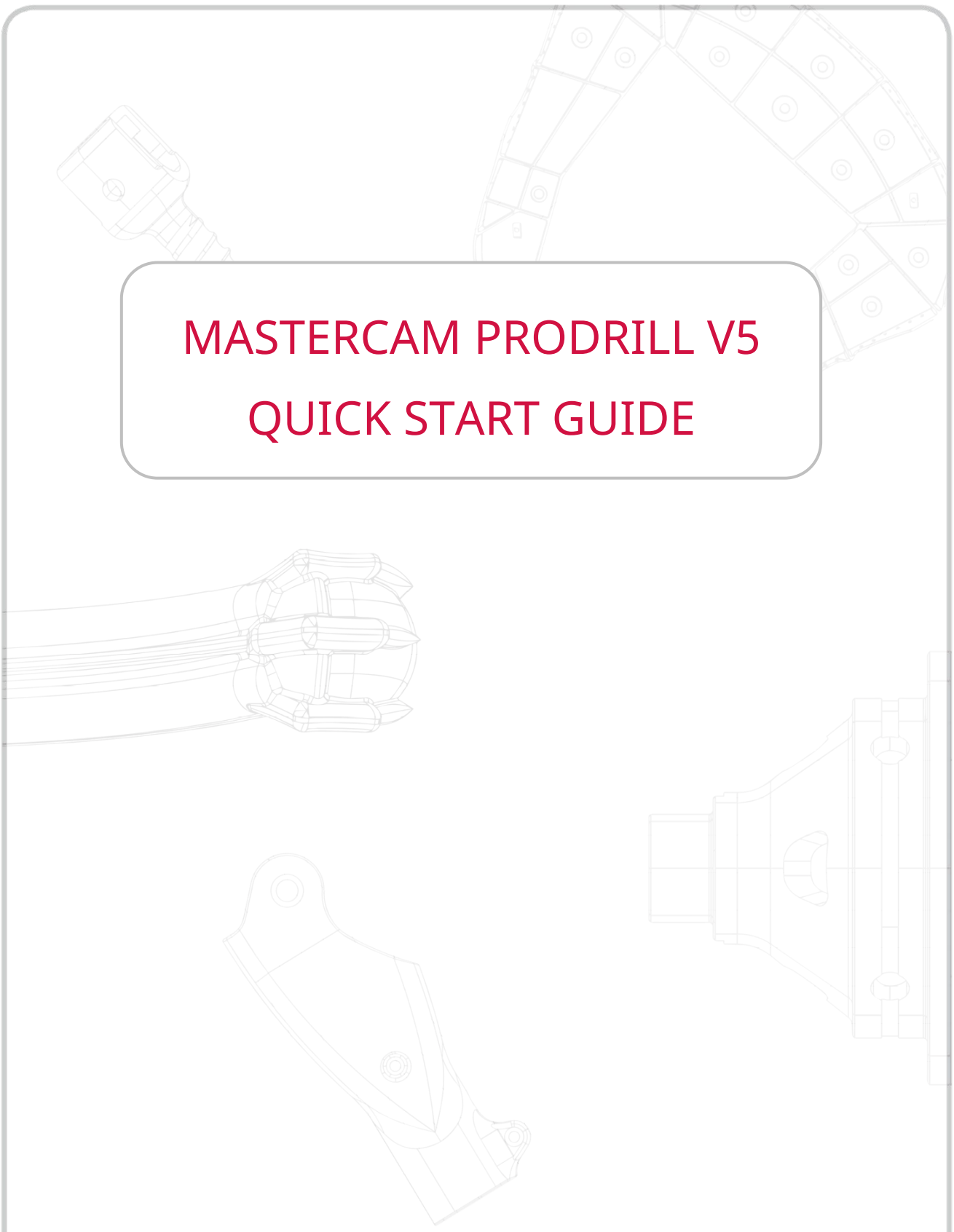
MASTERCAM PRODRILL V5
QUICK START GUIDE



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam®
Be Dynamic.

Revised 10/20/2021

CONTENTS

INTRODUCTION 4
PREREQUISITIES 4
SOFTWARE 4
RIGHT MOUSE CLICK OPTIONS 4
DOCUMENTATION 6
PROCESSING A PART IN PRODRILL V5 7
DIAMETER RANGE 7
ANALYZING THE PART FOR FEATURES 10
APPLYING PROCESSES AND MACHINING A FORM 12
APLLYING DRILLING PROCESSES 13
MACHINING THE FORM 15
REPORTS - MACHINING TIME, DRILL LIST and TOOLS USED 16
MACHINING TIME REPORT 17
DRILL LIST REPORT 18
LIST OF TOOLS USED REPORT 18
DRILL FORMS THAT EXCEED MAXIMUM DRILL DEPTH 19
APPLYING PROCESSES BY CREATING TOOLS MANUALLY 21
CREATING A MANUAL PROCESS TO DRILL A FORM 25
MACHINING ONLY SELECTED POSITIONS BY CLONING FORMS 30
OPTION1: SELECTION FROM THE TREE 30
OPTION2: SELECTION FROM THE GRAPHICS 31
SORTING FORMS IDENTIFIED ON INCLINED PLANES 32
OPTION1: SORTING POSITIONS USING MANUAL SORT METHOD 33
OPTION2: SORTING POSITIONS USING AXIS SORTING METHODS 35
SPLIT FORMS TO BE DRILLED FROM OPPOSITE PLANES 36
SETTING TAPPED AND REAMED FEATURES 38
SETTTING TAPPED FEATURES 39
SETTTING REAMED FEATURES 42
CREATING MULTIPLE PROCESSES, EDITING AND STORING TO LIBRARY 46
V5 LIBRARY 52

APPLYING PROCESSES FROM THE LIBRARY 55
SPLIT CHAMFERS ON OPPOSITE PLANES AND MACHINE 56
SPLITTING THE CHAMFERS TO OPPOSITE SIDE 58
APPLYING PROCESSES AND STORING TO LIBRARY 59
MODIFYING PROCESS NAMES AND FORMS NAME IN LIBRARY 63
REAPPLYING PROCESSES FROM LIBRARY 64
CREATING TOOLPATHS AND GROUPING BY VIEWS 65
SORTING FORMS GRAPHICALLY BY SKETCHING LINES 67
IMPORTING REAMED AND TAPPED FEATURES FROM A SOLIDWORKS FILE 70
IMPORTING COLORED FEATURES FROM A CATIA FILE 73
PRODRILL SUPPORT 76

INTRODUCTION

This guide will instruct you on how to use the various features of ProDrill V5 and automate the drilling of 2 to 5 axis parts in Mastercam.

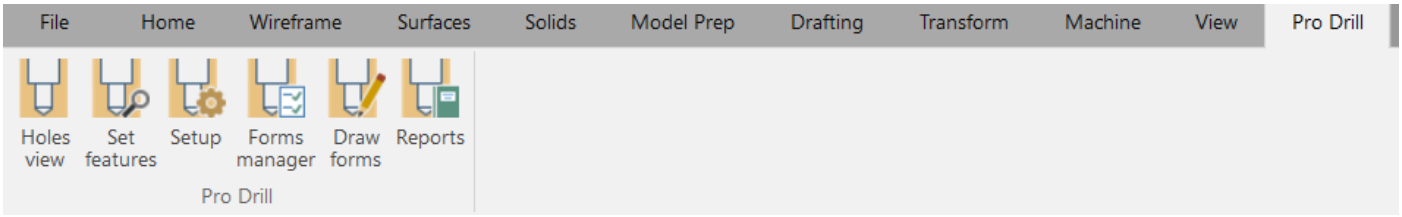
PREREQUISITIES

Please ensure the following items are installed and configured correctly before using this quick start guide.

SOFTWARE

- Mastercam installed and launched at least once before installing ProDrill.
- ProDrill V5 for Mastercam installation.

After installing, ProDrill will appear on its own ribbon tab in Mastercam.

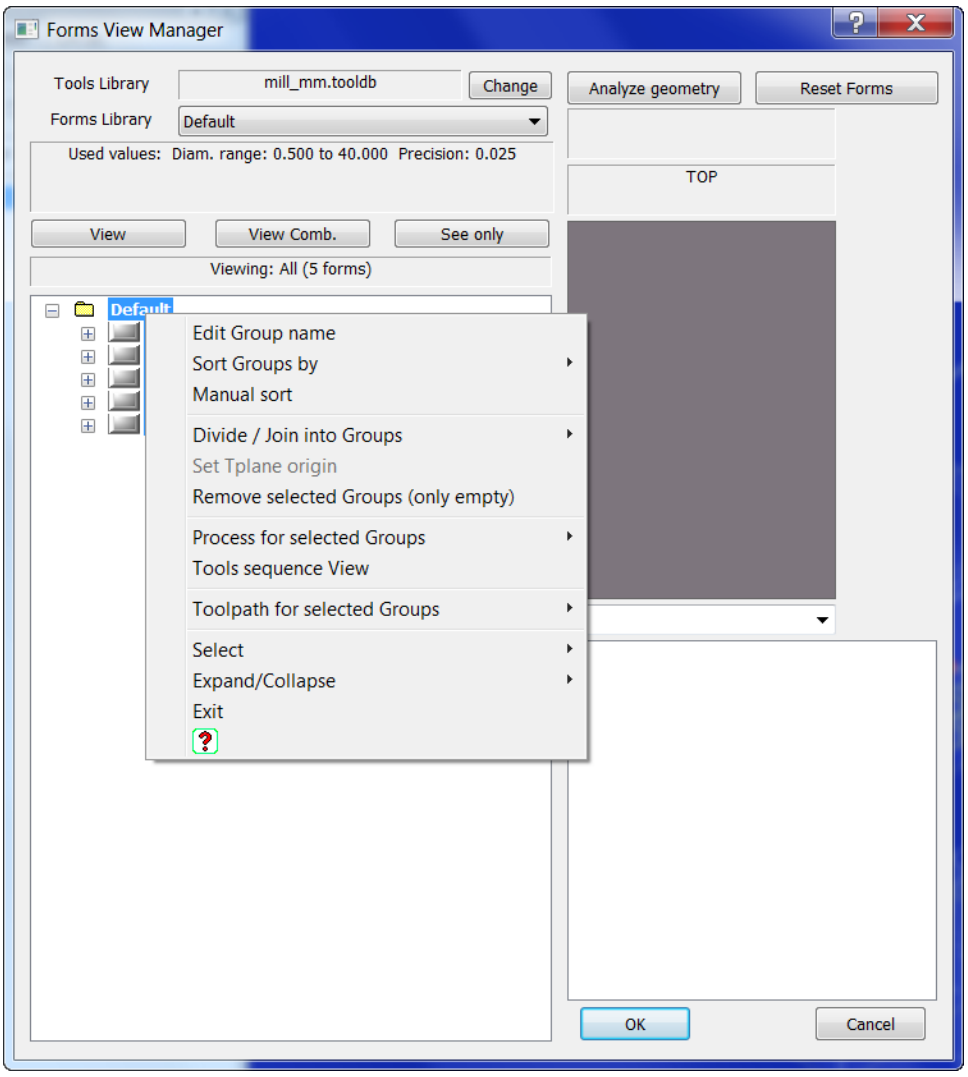


RIGHT MOUSE CLICK OPTIONS

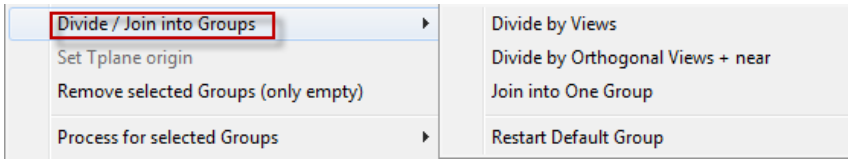
***The Right Mouse Click is one of the most important tools to optimally use ProDrill*.**

Forms View Manager

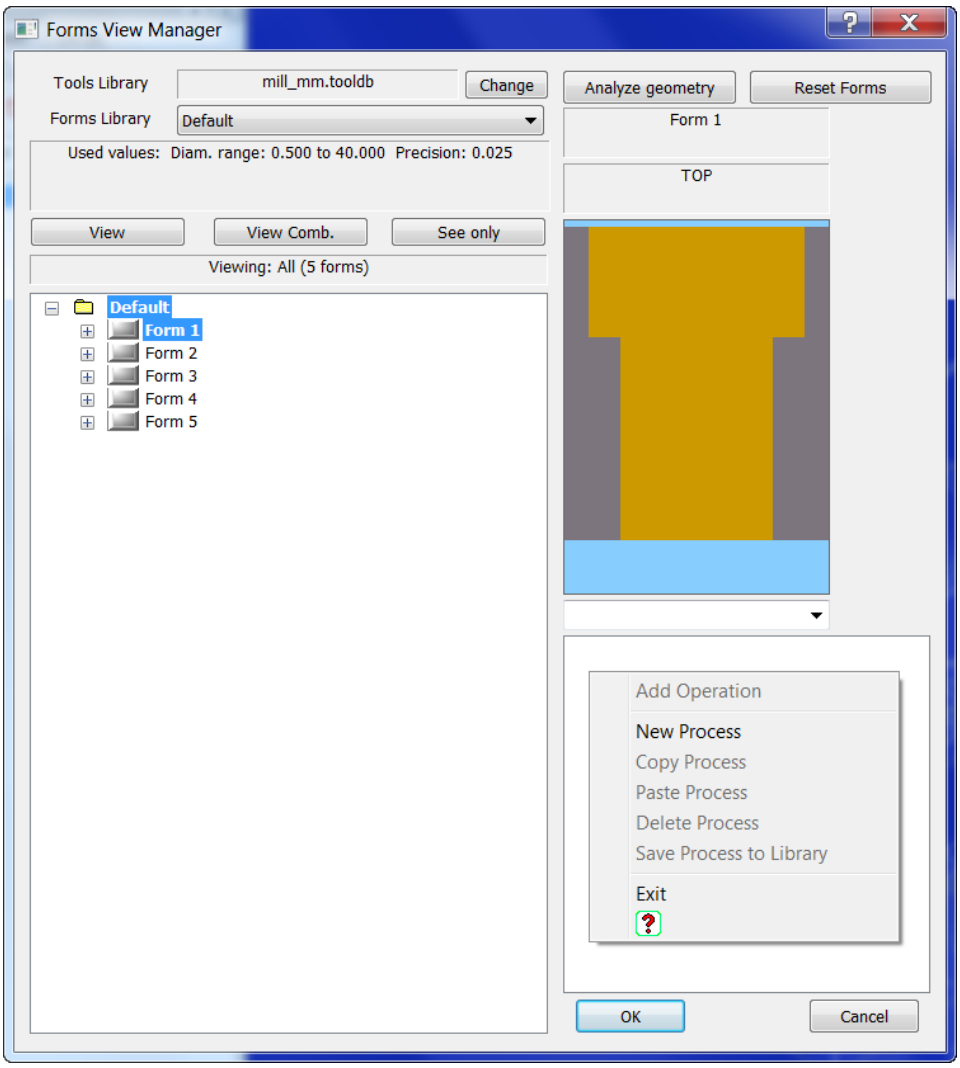
Right mouse clicking on **Default**, **Form** or **Position** will each present the appropriate list of actions that can be performed. Here, right mouse clicking on the **Default** group presents the following list of actions.



Sub menus are available for many options from the right click menu. Clicking on the arrow will display the appropriate actions available. For example, clicking on the **Divide/Join into Groups** offers the following options:



Right clicking in the **Process** dialog presents the appropriate list of actions also. If no processes are defined, you can create new processes for the selected Forms.



Remembering this can take you a long way towards quickly learning ProDrill!

DOCUMENTATION

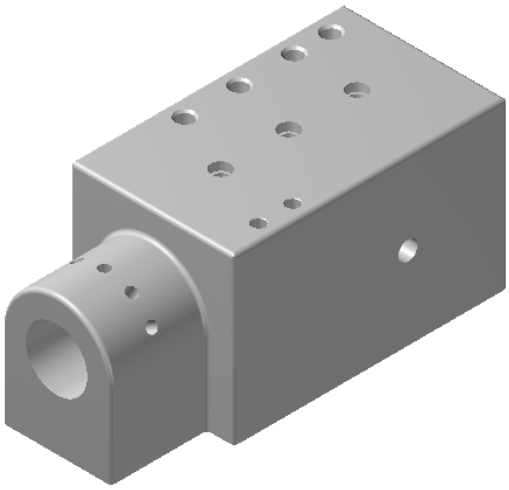
Context sensitive help is offered for most of the dialog settings. **Online Help** is available **for** all **dialogs** and **right click menus**. The Online Help file is stored to the \Mastercam\help\language folder upon installation. Tooltips are also available for most settings.

Other ProDrill documentation is added to the **\Mastercam\Documentation\Language** folder in the form of PDF documents.

PROCESSING A PART IN PRODRILL V5

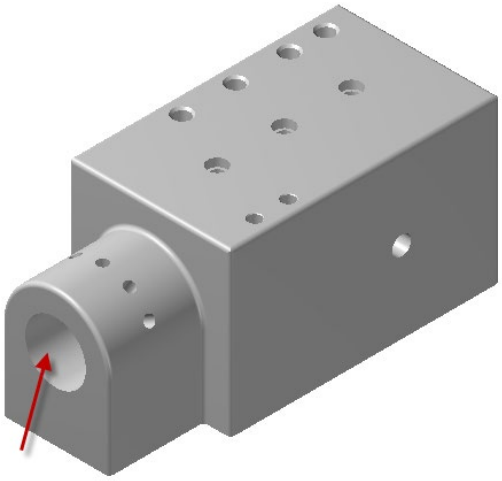
We will review various features offered in ProDrill V5 using an example part provided with the installation.

Load the file **PART_4.MCAM** from the **C:\Users\Public\Public Documents\Shared Mastercam 20XX\ ProDrill_V5\ProDrill_Parts** folder. This is a Solid part.

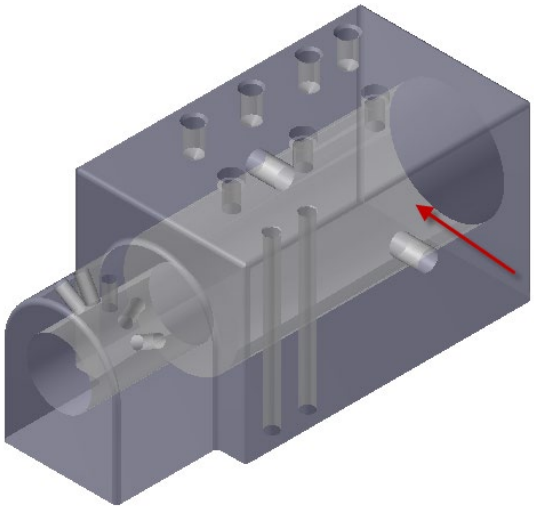


DIAMETER RANGE

ProDrill's advanced identification algorithms will automatically locate all possible drilled features. You will need to set the minimum and maximum diameter to be identified. For the example above, we can assume that the center cylinders have already been machined and should be excluded when using ProDrill. To know the maximum diameter to be machined by ProDrill use the Mastercam *Analyze, Dynamic* function to analyze the radius by clicking on the inside wall of the cylinder as shown in the diagram below. At the bottom of the Mastercam screen a *Radius of Curvature* value of 22.5mm (45mm diameter) will be shown.

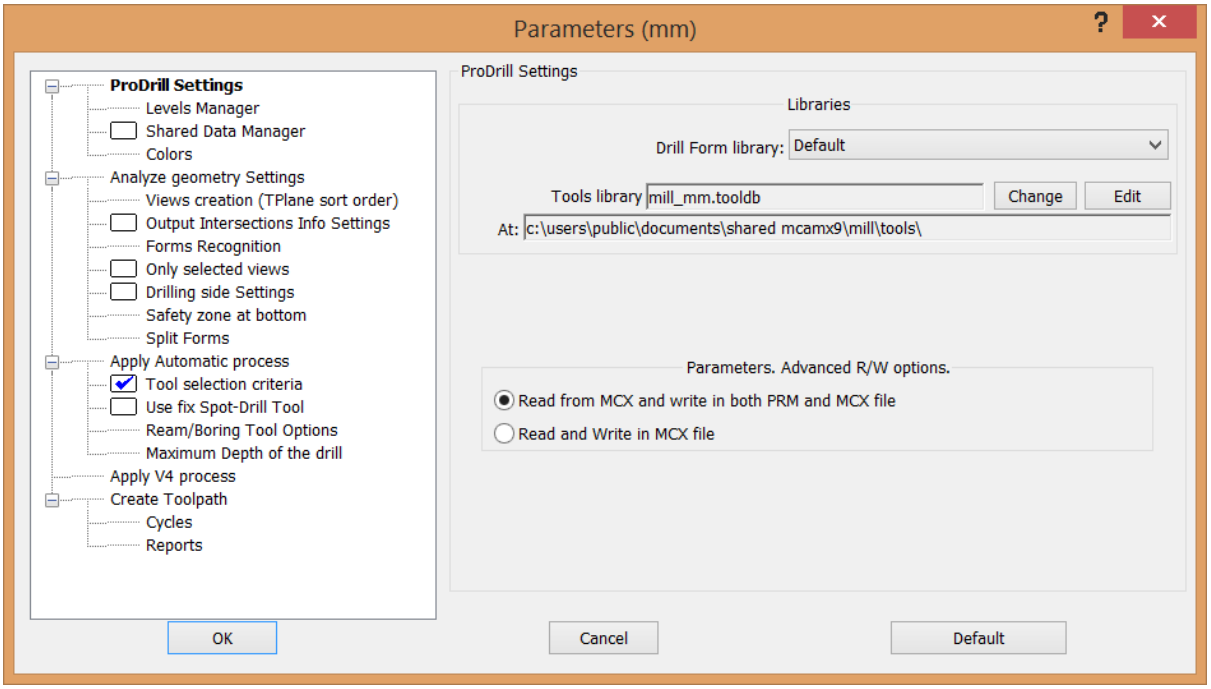


Since the other end of the selected cylinder is larger (80mm diameter) and previously machined no additional analysis is required.

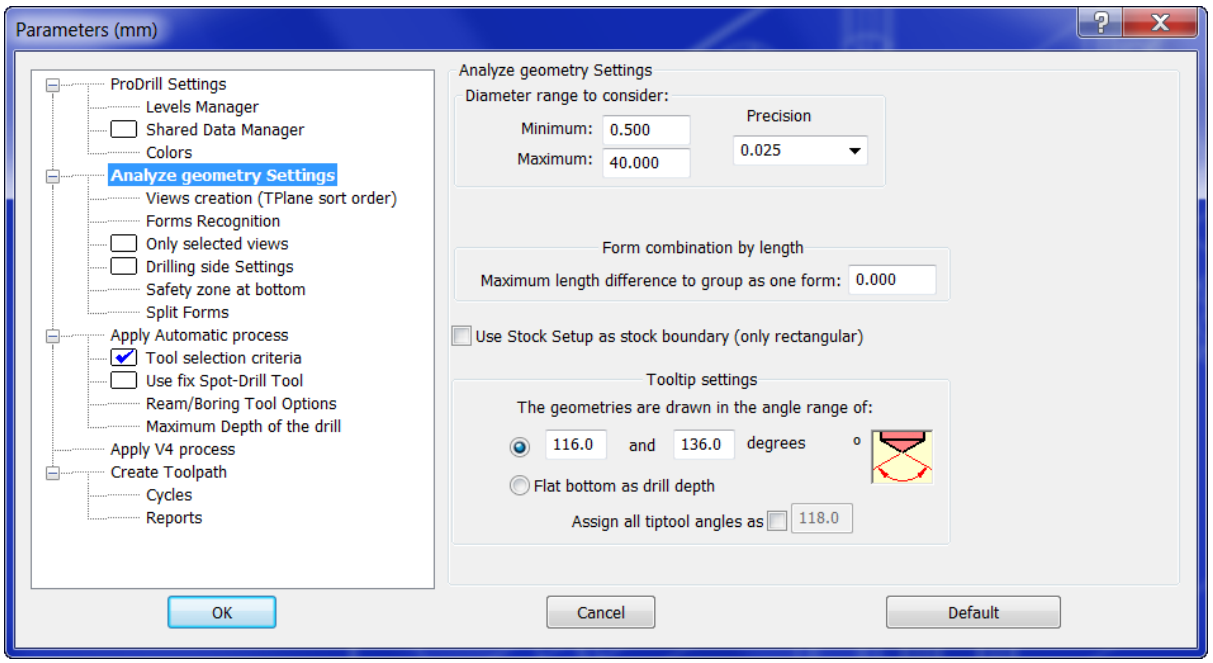


Once you have determined what the maximum diameter should be (less than 45mm) click on the Setup icon .

Ensure that the tool library settings are set as shown below:



Set the values in the parameter screen for **Analyze geometry Settings** Tab as shown below. Diameter range should be se to a maximum of 40mm diameter.

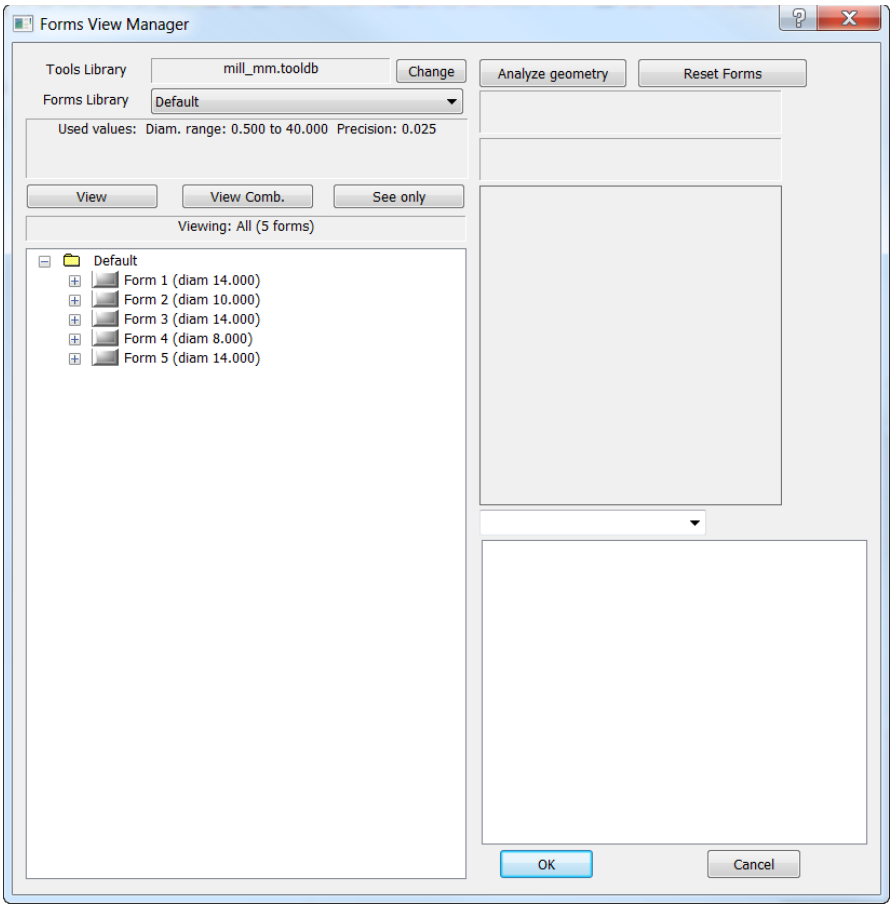


Leave the rest of the tabs with the default settings.

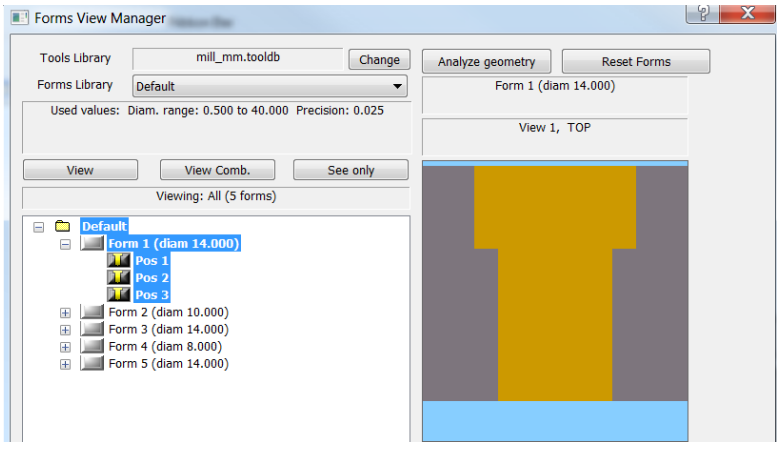
Clicking on the **Default** button will also reset the settings to ProDrill defaults. Most of the default settings can be controlled from the ProDrill_V5.TXT file under the [Default] section.

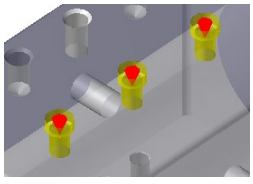
ANALYZING THE PART FOR FEATURES

To start analyzing the part, click on the icon. ProDrill will automatically and rapidly analyze the geometry and present the resultant display in the following dialog.

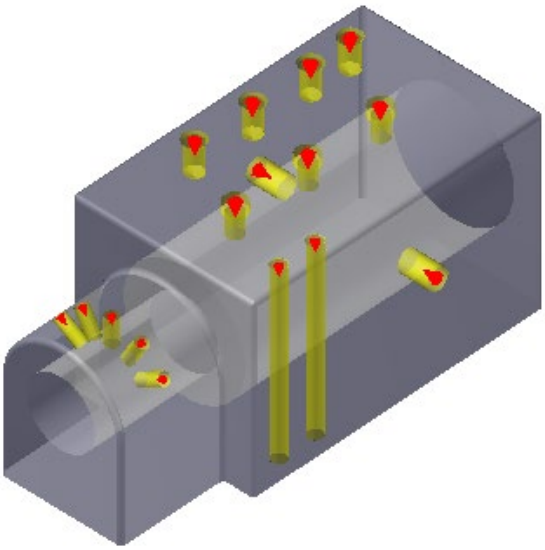


All the forms are identified on the part within the set diameter range. No toolpath operations are added to any of the forms. Clicking on any of the forms displays a graphics of the current form. **Highlighting a cylinder also displays the dimensions of the cylinder including the total depth at the corresponding cylinder**. When a Form is selected from the tree (in the case below, highlight Form1), the corresponding geometry is also highlighted.

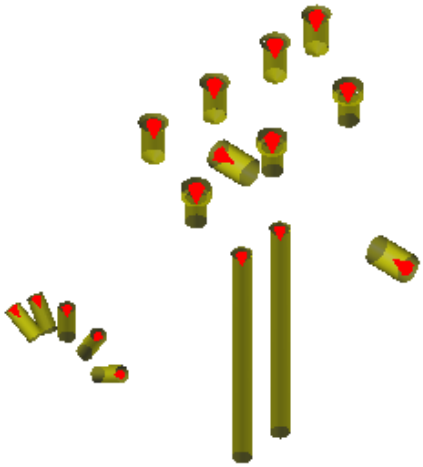




Highlight the **Default** group as displayed. All the drill forms encountered on all the planes are highlighted in the Mastercam select color and displayed. Red pointers indicate the drilling direction.



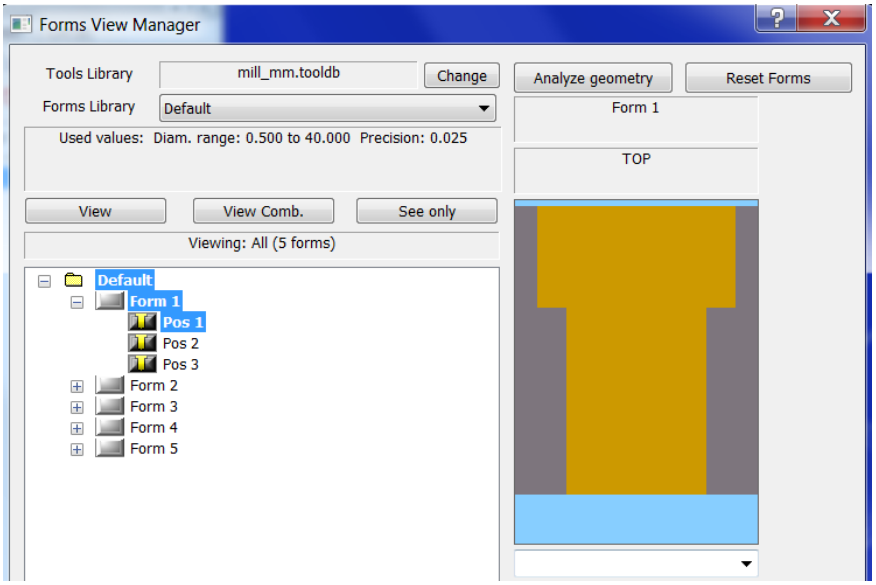
Click on **See Only** from the dialog to see the drill forms only.



In a few seconds, you have identified all the drill forms encountered and verified the forms encountered!

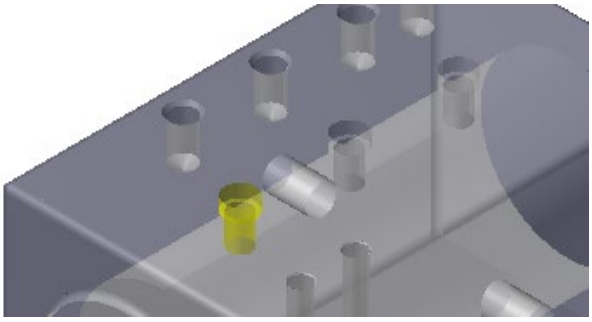
APPLYING PROCESSES AND MACHINING A FORM

ProDrill V5 identifies all the drill forms found in the part and sorts them by form type and then lists all the positions found for each Form type. The positions are listed under the form type and are independent of the tool plane they are identified on.



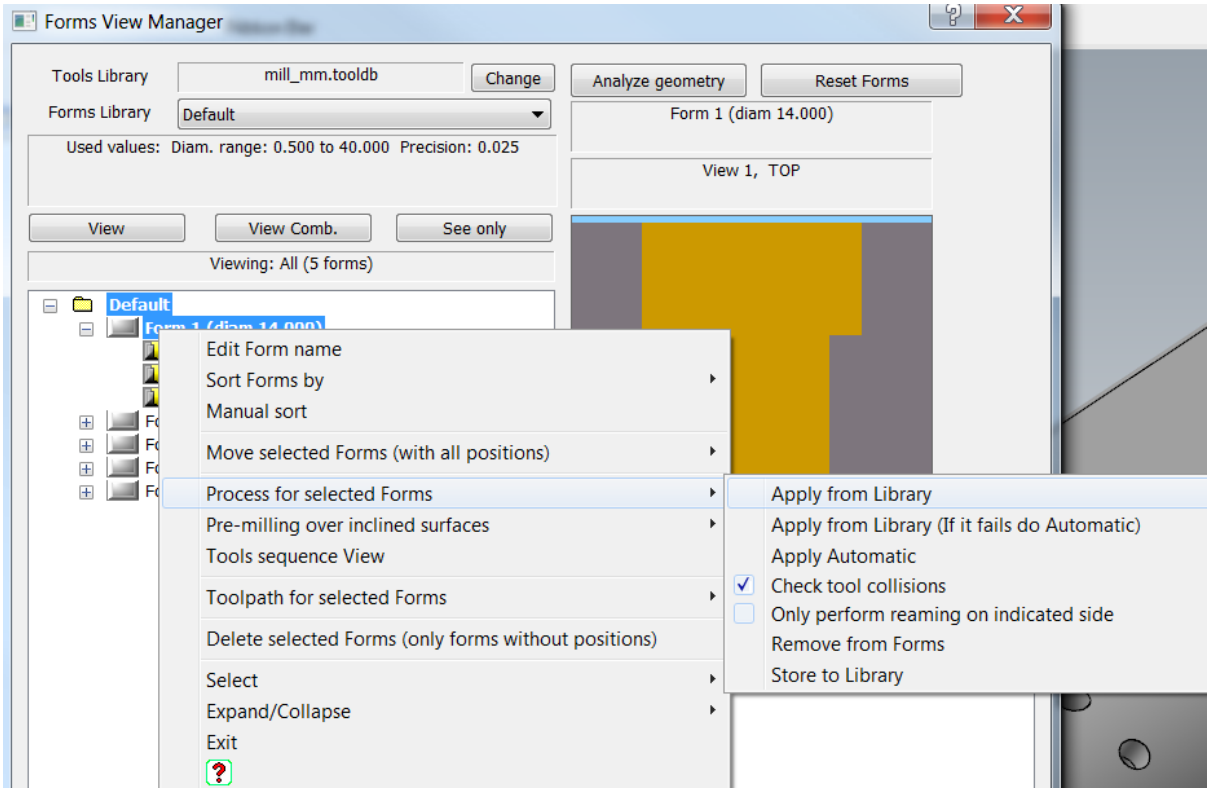
In the above case, we see that **Form 1** has 3 different positions. The image of Form 1, in this case a counter bored hole is displayed to the right of the form.

The graphics area also displays the appropriate item selected. In this case, only **Pos 1** is selected, therefore only 1 counter bore hole is displayed.



APLLYING DRILLING PROCESSES

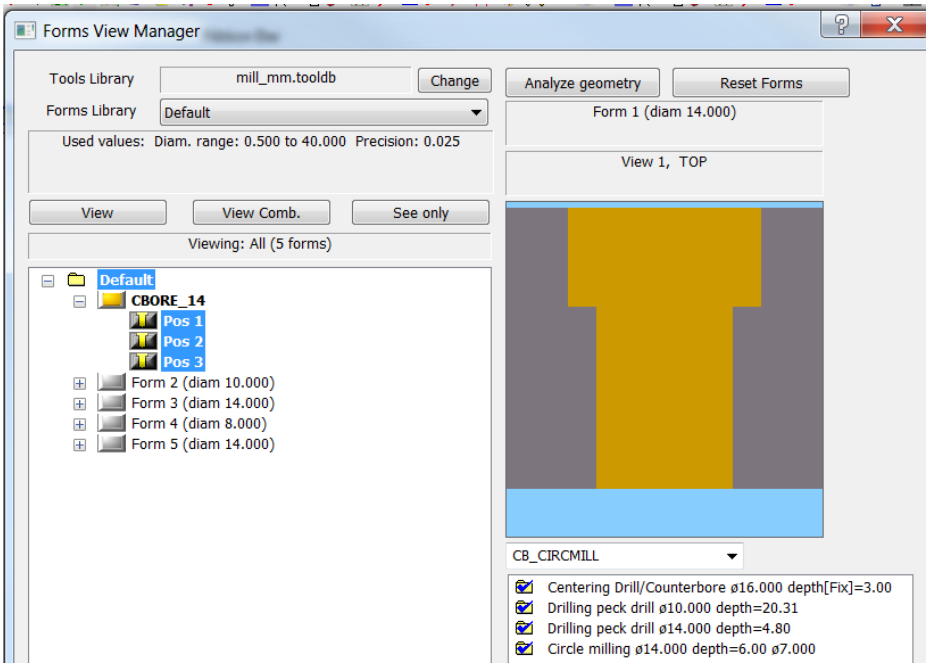
If we wish to machine the 3 positions in the form, **click on** the **Form 1** name to highlight all 3 positions. Then **right click** and choose **Process for selected Forms**, **Apply from Library**.

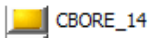


ProDrill searches for the drilling process in the new V5 Library. If it does not exist, it will prompt the user to use the Automatic Forms Process Manager (from Version 3) to create the drilling processes.

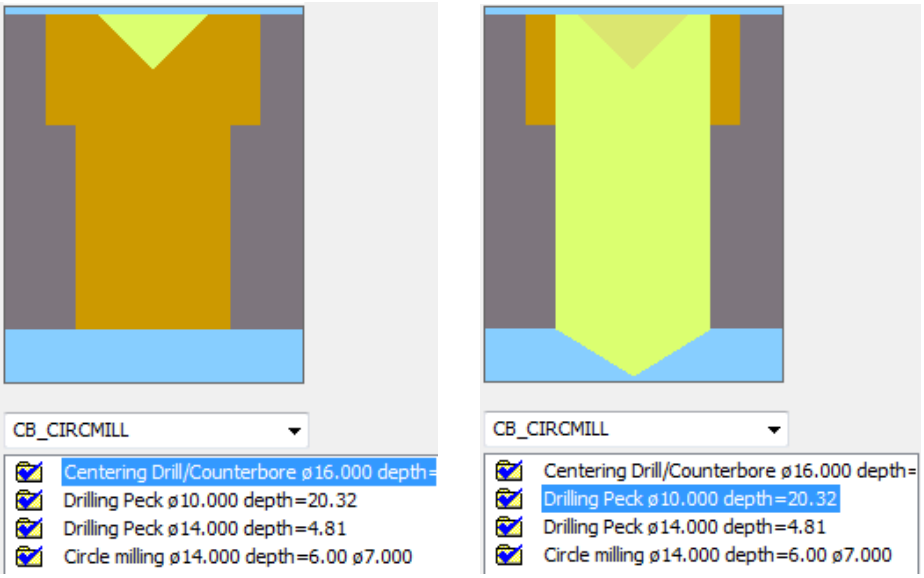
The Automatic Process method will create a set of drilling operations by choosing the relevant tools (creating them in the process if they do not exist). The user can then verify the processes to ensure that these are the desired operations or modify them as needed.

The following is the result:



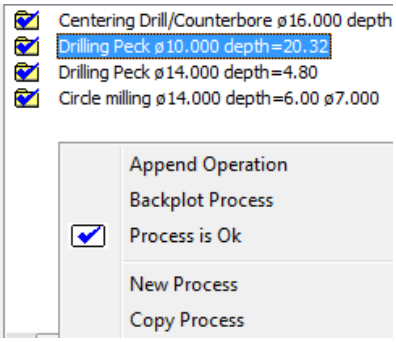
An orange icon  along with the name of the form is created in place of Form 1. This is to indicate that the processes have been added but they still need to be verified by the user!

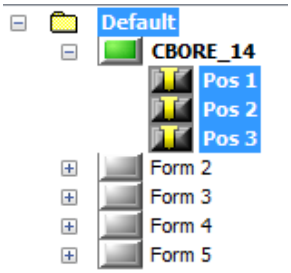
On the right side, in the process dialog, 4 different drilling operations are created. You can do a quick visual backplot of the drilling processes by clicking on the appropriate operation. The graphic image is updated with the relevant tool. Note that the previous tool is also visual in the graphic to indicate the depths and areas machined with the tools!



This is a quick way to verify the drilling operations generated for processing this part.

Once the processes have been verified and you wish to approve the process, right click within the Process area and click on **Process is OK**. You will now notice that the icon turns green to indicate that the user has verified the processes and accepts it.

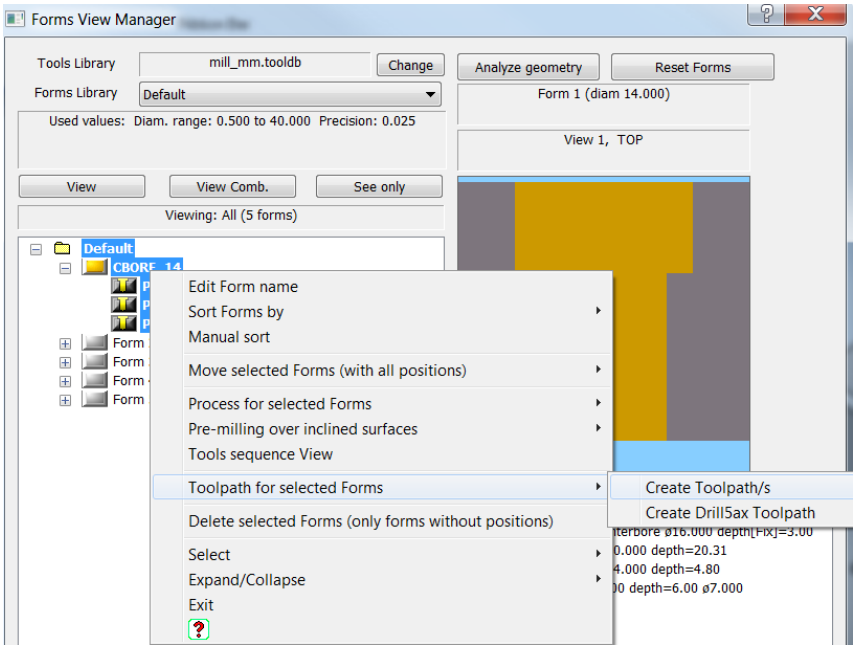




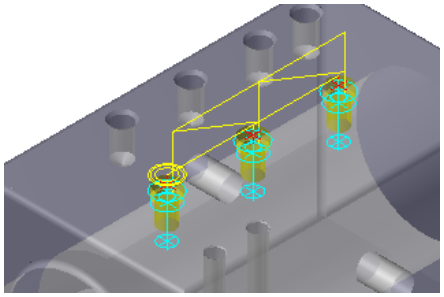
You have now applied machining processes, verified the process and approved it.

MACHINING THE FORM

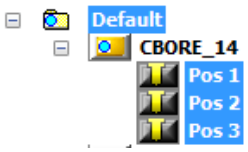
To machine the form, right click on the Default name or on the form itself and select **Toolpath for selected Forms**, **Create Toolpath/s**.



Mastercam toolpaths are automatically created for the selected form on the toolplane that the forms were identified on. In this case, it is the Top construction plane.



The icon also changes to the following to indicate that toolpaths were created for this form. Note that the folder group for the form as well as the Default group have a blue dot to indicate that toolpath/s exist.



You can now proceed to create toolpaths for other forms using the same manner as described!

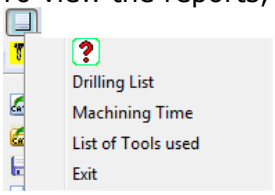
REPORTS - MACHINING TIME, DRILL LIST AND TOOLS USED

ProDrill generates a machining time report and drill list report when you create toolpath operations in ASCII format. The list of tools used report is generated when the option is selected in the menu. The files can be accessed in the same folder as the MASTERCAM file.

For example, part name PART_4.MCAM

Machining time report file: PART_4**_MT**.TXT
Drilling list report file: PART_4.TXT
Tools used report file: PART_4**_TOOLS**.TXT

To view the reports, click on the Reports icon. Choose the corresponding option from the list:



Note that the **List of Tools used** report is not generated till you select the option from the menu above.

MACHINING TIME REPORT

ProDrill generates a Machining Time Report for all the toolpaths processed. The feeds and speeds depend on your Machine Group - Tool Settings and the Rapid Rate and Tool change depend on the default Post Processor used.

From the **Reports** icon, click on **Machining Time**. The corresponding file (in this case, the PART_4_MT.TXT) is opened using the default editor in Mastercam.

```
    ************************************
         Machining time calculation
    ************************************

Part: C:\USERS\PUBLIC\PUBLIC DOCUMENTS\SHARED MASTERCAM 20XX\PRODRILL_V5\PRODRILL_PARTS\PART_4

Time computed in minutes.

*** Operations group Default ***

OPERATION Drill/Counterbore - Centering Drill/Counterbore ø16.000 depth=3.00
     TOOL T8  SPOT_DRILL Diameter 16.00  - ** 16.00 x 19.00 SPOT_DRILL
 GEOMETRY 3   points
     FEED 171.8 Mm/min (0.150 Mm/Rev)
      RPM 1145
     TIME 0.09

OPERATION Peck - Drilling Peck ø10.000 depth=20.32
     TOOL T110 DRILL Diameter 10.00  -  10. DRILL
 GEOMETRY 3   points
     FEED 137.4 Mm/min (0.120 Mm/Rev)
      RPM 1145
     TIME 1.00

OPERATION Peck - Drilling Peck ø14.000 depth=4.80
     TOOL T150 DRILL Diameter 14.00  -  14. DRILL
 GEOMETRY 3   points
     FEED 220.9 Mm/min (0.180 Mm/Rev)
      RPM 1227
     TIME 0.21

Group time:
  3 Tool changes: 0.05
      Rapid time: 0.00
  Machining time: 1.30
      Total time: 1.35

************************************
Results:

  3 Tool changes:   0.05
      Rapid time:   0.00
  Machining time:   1.30

************************************
      Total time:   1.35 minutes
************************************
```

DRILL LIST REPORT

When the toolpaths are generated, the Drilling List is also generated for the listed operations. The Drill list is handy for shops that require a listing of all the operations in an ASCII format for manually performing the operations OR for CMM inspection

From the **Reports** icon, click on **Drilling List**. The corresponding file (in this case, the PART_4.TXT) is opened using the default editor in Mastercam.

You will notice that all operations are listed. Those that will be machined on the CNC will be marked with <<<< *Created on CNC machine* >>>> next to the Operation name. (In this case, all operations will be listed as created on the CNC machine).

```
********************************************************************************
                            TOP
********************************************************************************

-------------------------------------
Operation: Centering Drill/Counterbore ø16.000 depth=3.00 <<<< Drilled on CNC machine >>>>

** 16.00 x 19.00 SPOT_DRILL  Diameter: 16.00

     X        Y        Depth
    -0.      -50.      -3.   Blind
    -0.       -0.      -3.   Blind
     0.       50.      -3.   Blind

-------------------------------------
Operation: Drilling Peck ø10.000 depth=20.32 <<<< Drilled on CNC machine >>>>

 10. DRILL Diameter: 10.00

     X        Y        Depth
    -0.      -50.      -20.32 Through Hole
    -0.       -0.      -20.32 Through Hole
     0.       50.      -20.32 Through Hole

-------------------------------------
Operation: Drilling Peck ø14.000 depth=4.80 <<<< Drilled on CNC machine >>>>

 14. DRILL Diameter: 14.00

     X        Y        Depth
    -0.      -50.      -4.8  Blind
    -0.       -0.      -4.8  Blind
     0.       50.      -4.8  Blind
```

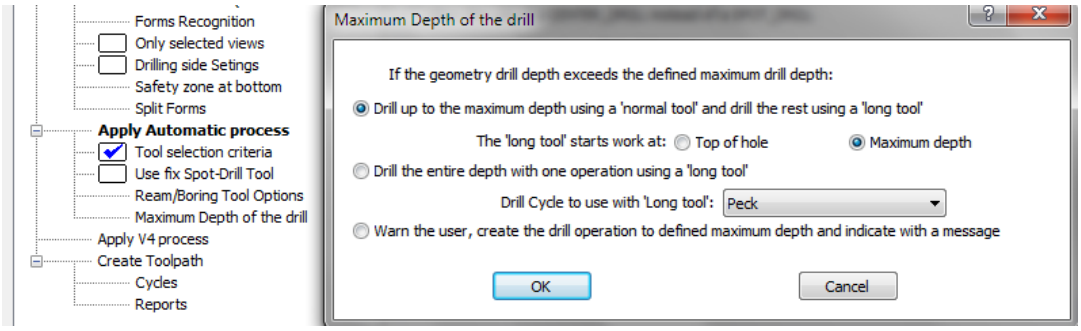
LIST OF TOOLS USED REPORT

From the **Reports** icon, click on **List of Tools Used**. The corresponding report is generated and the file (in this case, the PART_4_TOOLS.TXT) is opened using the default editor in Mastercam.

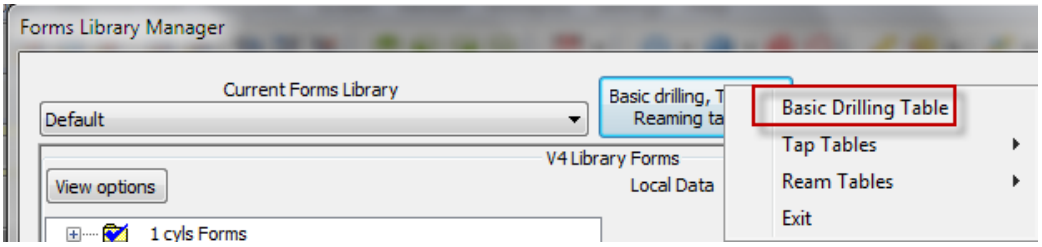
```
Used  Diam            Name              Num
 OP  14.000  14. DRILL                  150
 OP  16.000 ** 16.00 x 19.00 SPOT_DRILL       8
 OP   7.000    7. FLAT ENDMILL            216
 OP  10.000  10. DRILL                  110
```

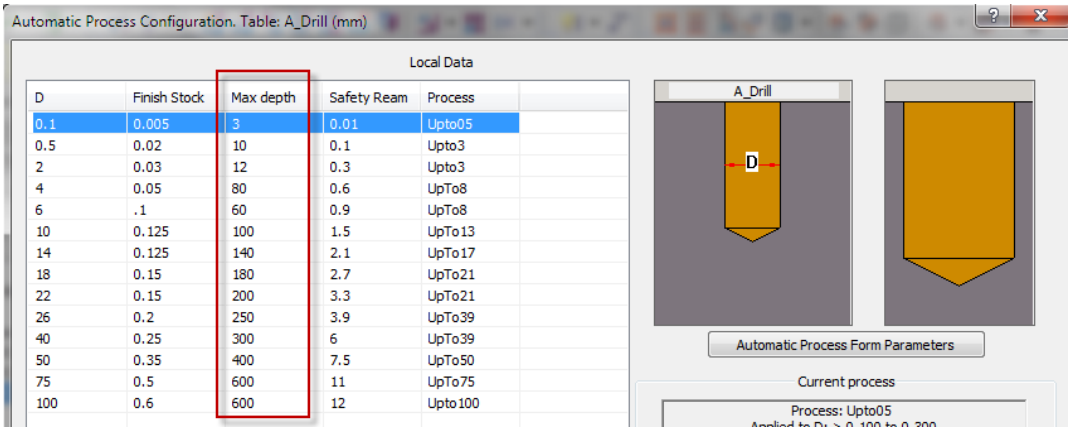
DRILL FORMS THAT EXCEED MAXIMUM DRILL DEPTH

Prodrill automatically identifies forms that exceed maximum drill depth. When you apply a drilling process, it splits the drill form based on the following settings:

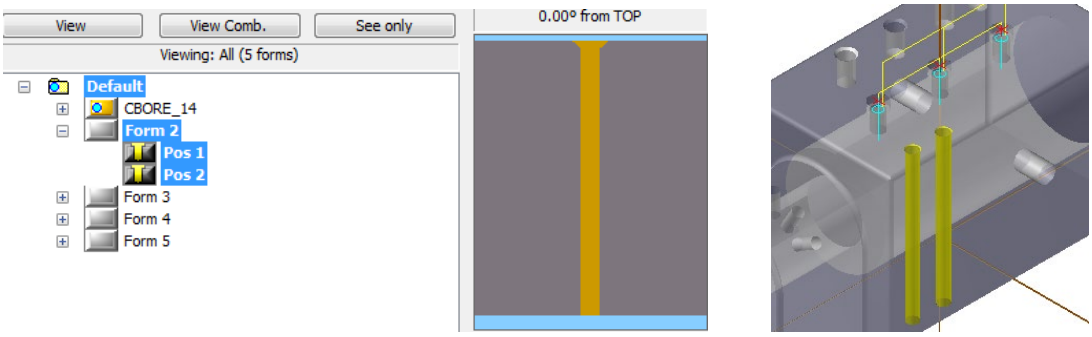


Leave the settings as shown above. The maximum depth value per diameter is set in the **A_Drill** table of the automatic processes. You can access this table from the **Forms Manager.**

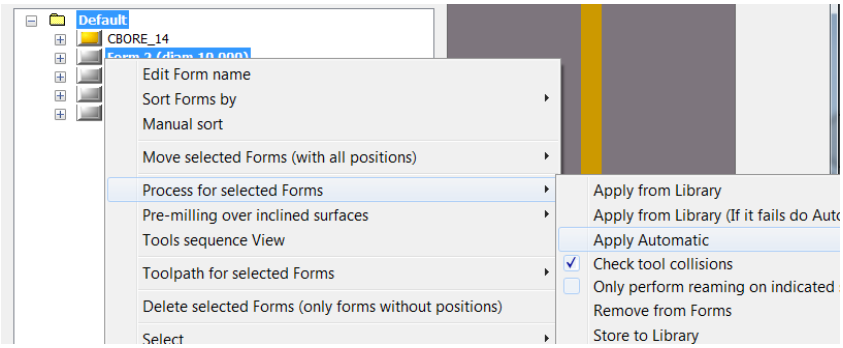




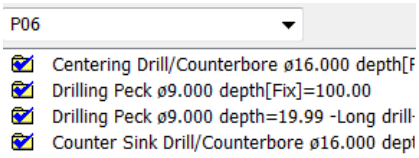
In the current part, there is one form that exceeds maximum depth (Form 2).



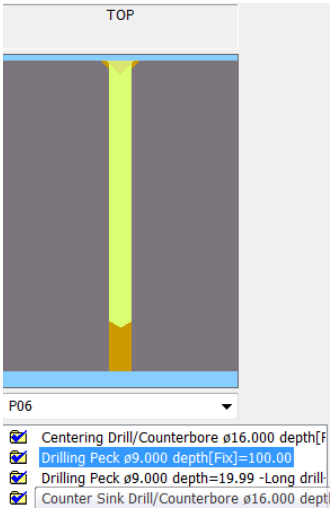
Apply an **Automatic Process** to the selected **Form 2**.

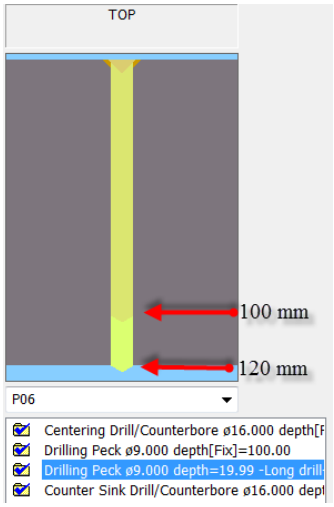


The following drilling processes are added. Note that the drilling processes are split into 2 operations. One operation that drills to a maximum fixed depth of 100mm and another that drills to the remaining depth.



Upon clicking on the operations, you will notice that the drilling operation 2 stops at 100mm and the next operation drills the remaining depth. The drill start location for the deep hole is controlled in the Settings tab. In this case, it will rapid to a few millimeters above the previous operation and then start drilling.



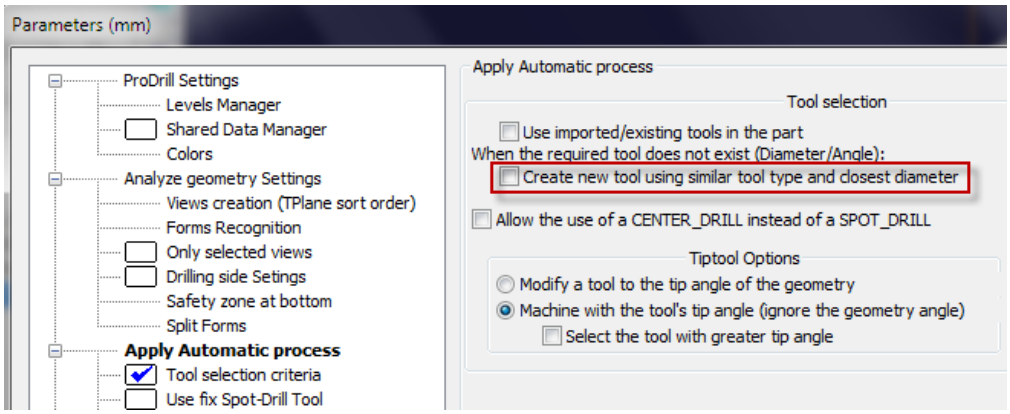


Creating the toolpaths will generate the corresponding drilling operations.

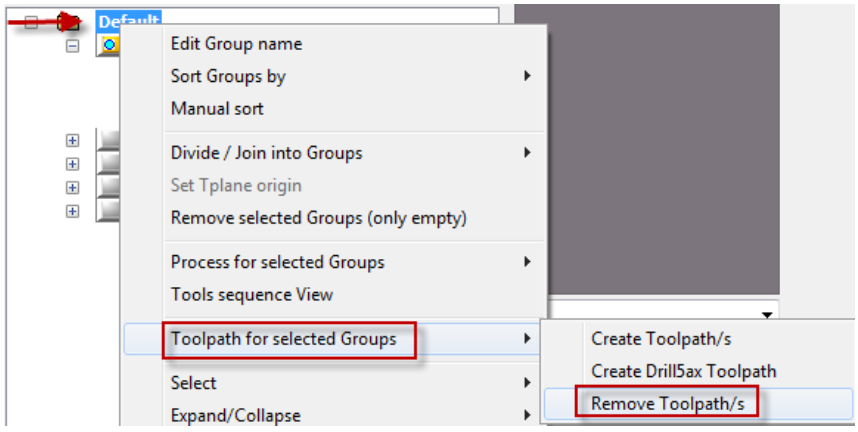
APPLYING PROCESSES BY CREATING TOOLS MANUALLY

ProDrill V5 allows the user optimal control on how tools are managed for machining the part. If new tools are needed, the user has the option to manually choose a tool from the library, create a new tool based on required specifications or customize an existing or newly created tool.

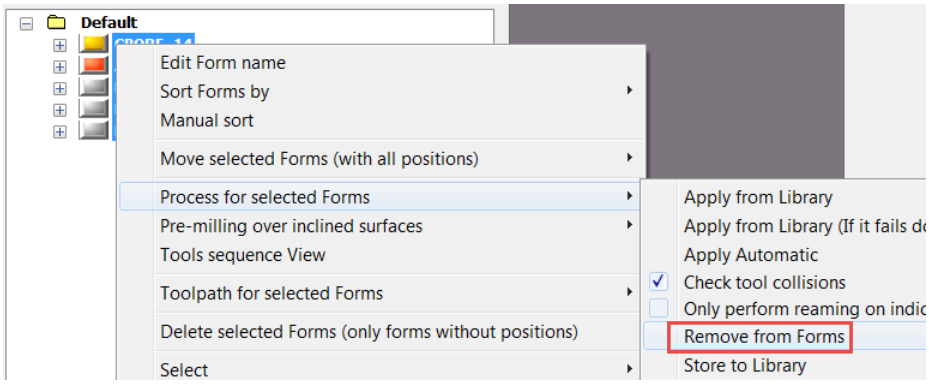
If the user does not wish Prodrill to create the new tools automatically when applying Automatic processes (note that V5 Library processes already have tools associated to the process through a separate tool library), the following parameter setting needs to be changed. Uncheck the option to "Create new tool using…"



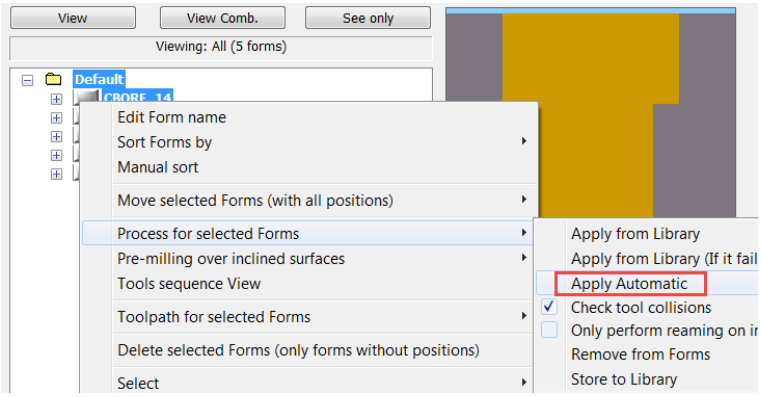
Launch the **Forms View Manager** and remove the toolpath generated first by doing the following - **highlight** the **Default** group to get the **Remove Toolpath/s** option:



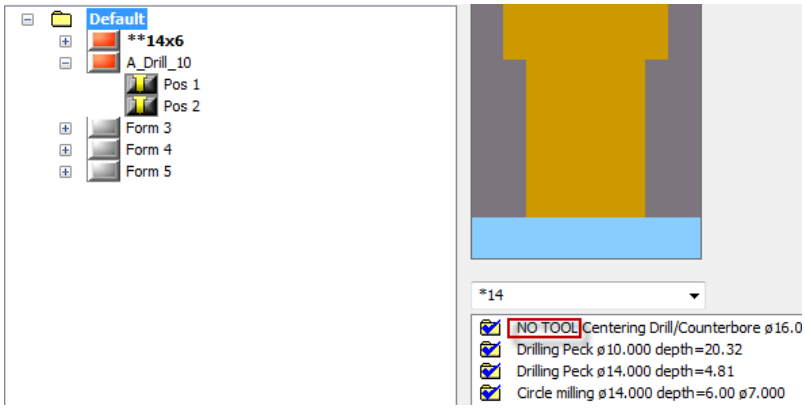
Next, remove the process applied to the form by doing the following:



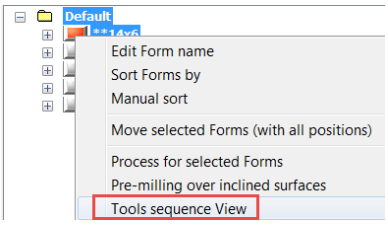
The part is now reset to the manner it was before applying processes to them. Now, apply a process again using the same menu and clicking on **Apply Automatic**.

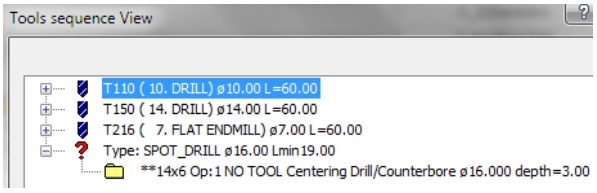


Any drilling process that does not have a tool in the tool library that matches the requisites for machining are identified in the list of operations as follows:

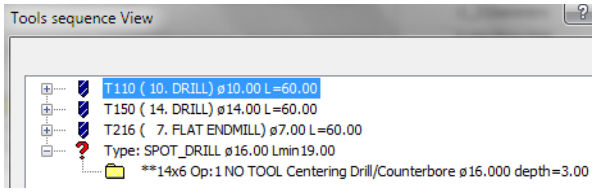


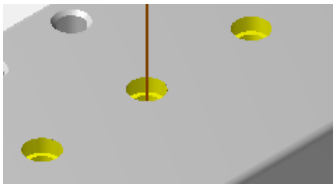
ProDrill did not create a tool for the centering as there was no tool available. In order to select or create a tool from the library, click on the **Tools sequence View** option from the same right click menu. The following dialog is opened:



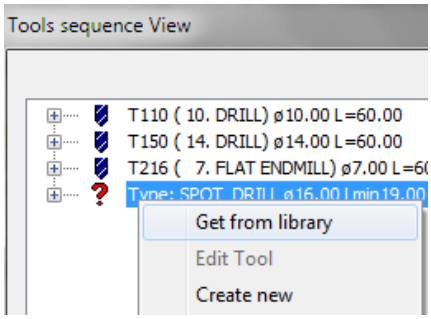


The tools found in the library are identified with the icon. The tool not found is identified by a ?. Clicking on any of the tools listed will also highlight all the forms and positions that will be drilled/machined with the tool:



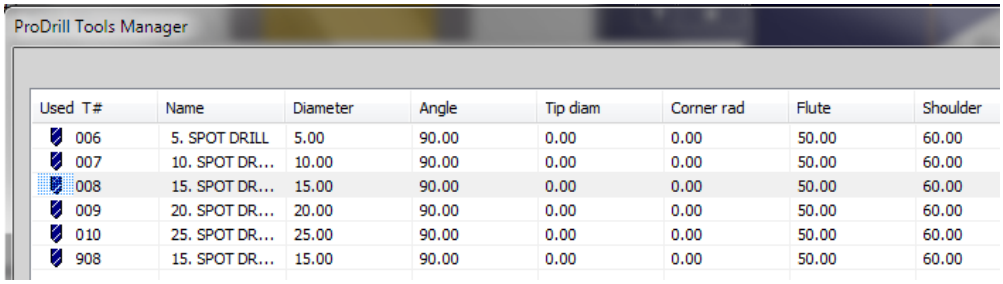


To assign a tool to the process, right click on the tool and select from the listed options:



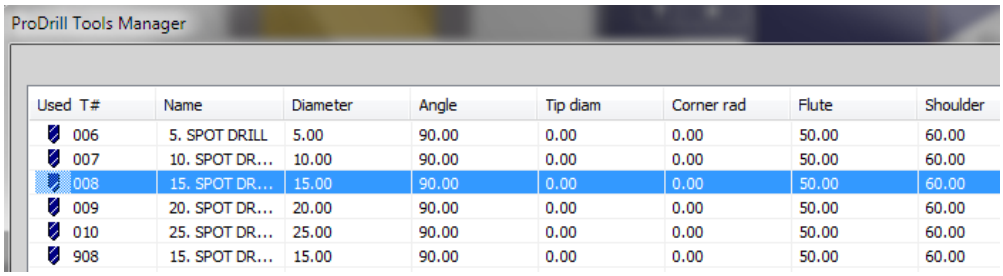
You can choose a tool from the library using "Get from library" or "Create new" tool. Since this is a spot/center drilling operation, we can choose a tool from the library itself. Select "**Get from library**".

ProDrill automatically scans the tool library and filters the available spot drill tools found in the library and presents it in a dialog. *The closest tool that matches the drill diameter (16mm in this case) is automatically highlighted as shown below*.



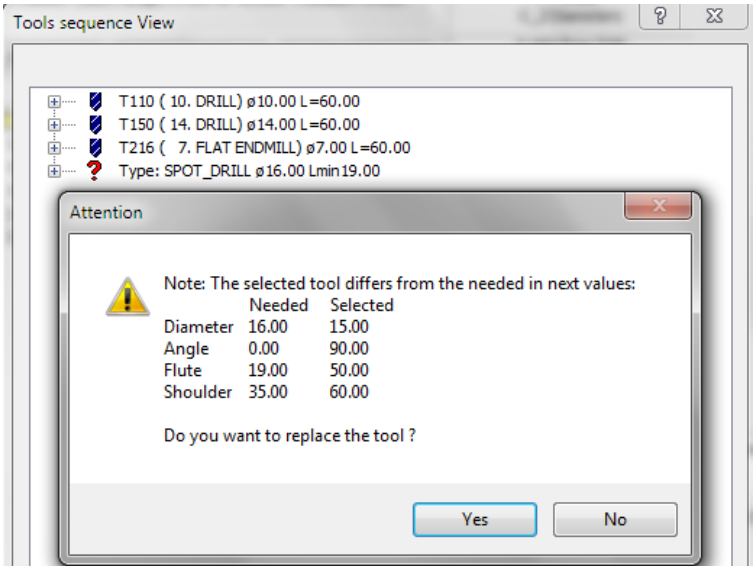
Used	T#	Name	Diameter	Angle	Tip diam	Corner rad	Flute	Shoulder
	006	5. SPOT DRILL	5.00	90.00	0.00	0.00	50.00	60.00
	007	10. SPOT DR...	10.00	90.00	0.00	0.00	50.00	60.00
	008	15. SPOT DR...	15.00	90.00	0.00	0.00	50.00	60.00
	009	20. SPOT DR...	20.00	90.00	0.00	0.00	50.00	60.00
	010	25. SPOT DR...	25.00	90.00	0.00	0.00	50.00	60.00
	908	15. SPOT DR...	15.00	90.00	0.00	0.00	50.00	60.00

ProDrill highlight the 15mm spot drill. If you right click on the tool, the Mastercam Edit tool dialog is opened to allow you to edit the tool if needed. Otherwise, click on the tool as indicated below and click on OK to select the tool.



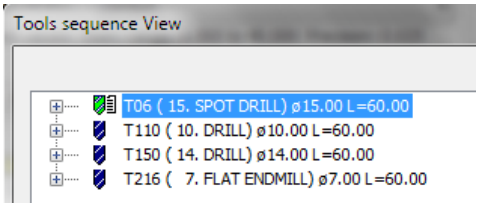
Used	T#	Name	Diameter	Angle	Tip diam	Corner rad	Flute	Shoulder
	006	5. SPOT DRILL	5.00	90.00	0.00	0.00	50.00	60.00
	007	10. SPOT DR...	10.00	90.00	0.00	0.00	50.00	60.00
	008	15. SPOT DR...	15.00	90.00	0.00	0.00	50.00	60.00
	009	20. SPOT DR...	20.00	90.00	0.00	0.00	50.00	60.00
	010	25. SPOT DR...	25.00	90.00	0.00	0.00	50.00	60.00
	908	15. SPOT DR...	15.00	90.00	0.00	0.00	50.00	60.00

Since we have selected a tool that does not precisely match the form machining requirements, the following warning is presented.



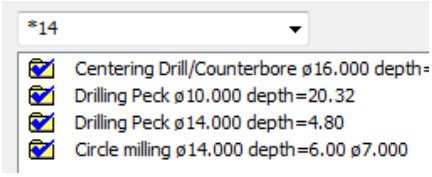
ProDrill intelligently verifies every step of the process definition and tool selection process to ensure that the right tool with the diameter and length are chosen to accomplish the task!

In this case, since this is a spot drilling operation, click on **Yes** to accept the tool selection.



The tool is now assigned to the process. Upon expanding the folder, you will notice the process that the tool will be used for!

Once you close the Tools Manager dialog, the process list is now updated and displayed as shown:



The message "No Tool" is now gone! You are now ready to create the toolpath as before.

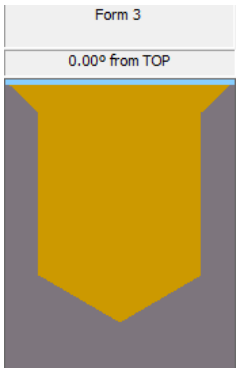
If there are other forms that require tools, the same logic can be applied.

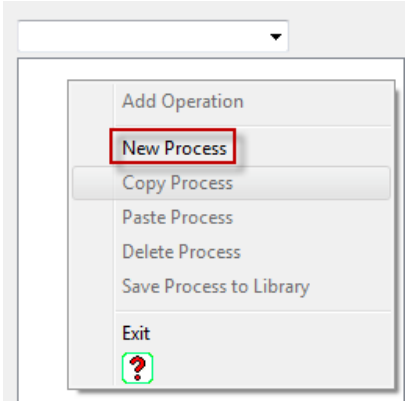
CREATING A MANUAL PROCESS TO DRILL A FORM

Creating new processes to machine forms is possible using the "Automatic process" method or from the stored "Library" method. If you prefer to create a process manually by defining each drilling operation, you can do that easily for a form.

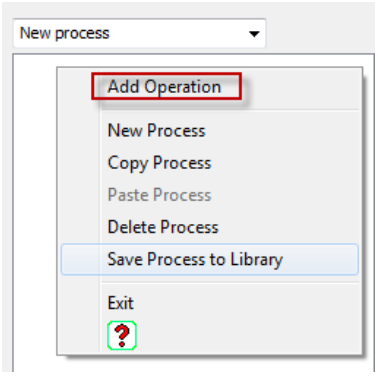
Important note: When you create manual processes and operations, tools can ONLY be selected from the available options. From the MCAM file, created manually or from the current library. ProDrill will not create any new tools automatically for you!

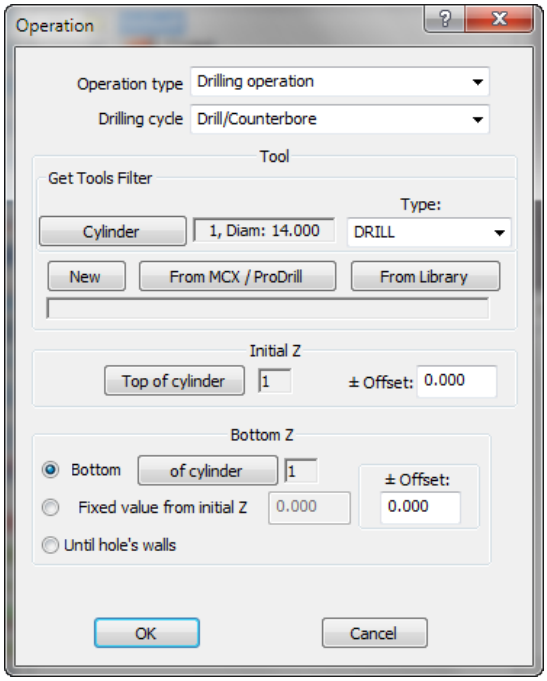
Choose **Form 3**. It is a drill form with a counter sink on the top. To create the drilling processes manually, right click in the process area and choose **New Process**.



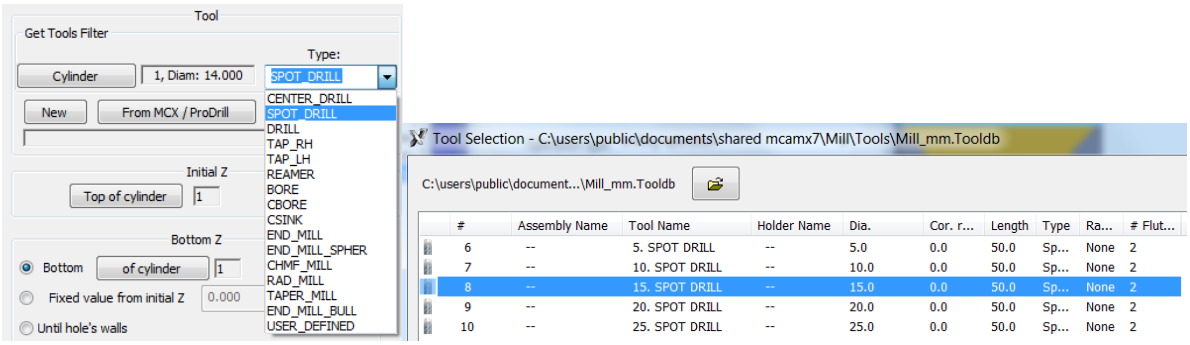


A process name called "New Process" is added to the list. Right click again and choose **Add Operation.** The **Edit operation** dialog is now open**.**

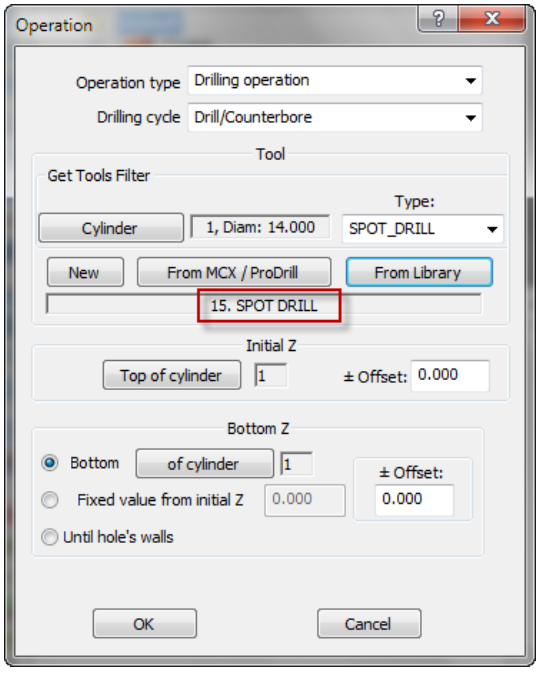


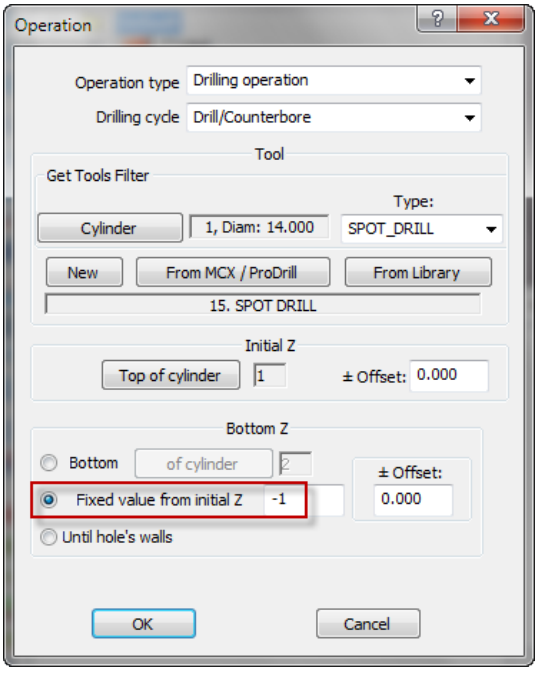


Using this dialog, we can define the first operation. The operation type and drilling cycle can remain the same. Choose a spot drill tool for this operation. The drill form diameter is 11mm. Click on **From Library** and choose a **Spot Drill** tool of **15mm** from the Tool library.

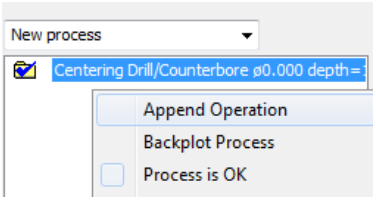


The following is the display after the tool selection. Now, you can set the spot drill depth. You can set this value as a fixed depth value. To do that, click on the "**Fixed value from initial Z**" and enter the value of **-1mm**. Make sure you set a negative value!

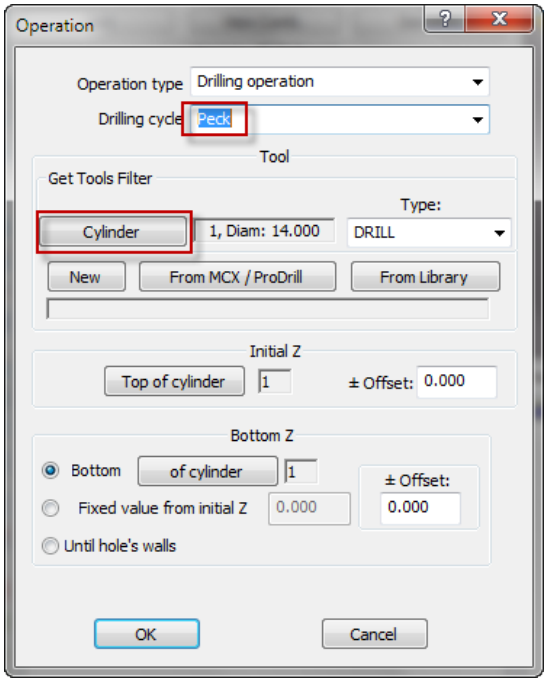


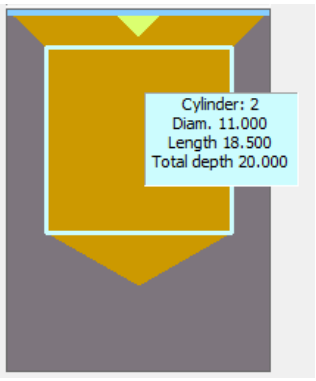


Click on **OK** to accept the first operation definition! If you click on the name of the operation that is listed, you will see the simulation on the graphic itself. You can now define the next drilling operation. Right click and choose "**Append operation**".

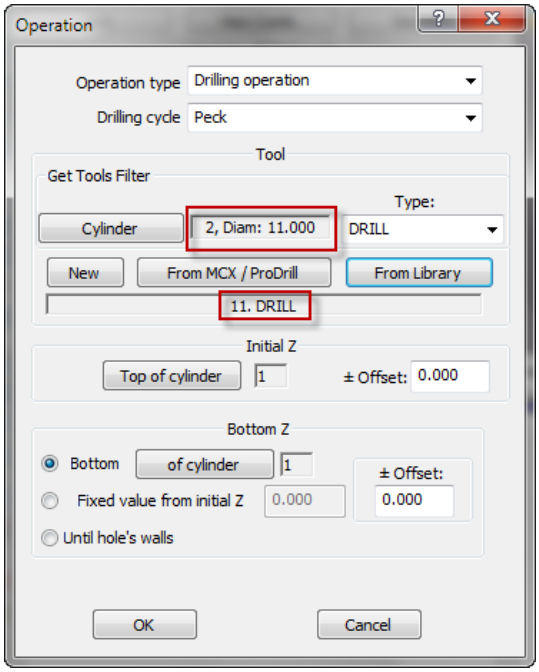


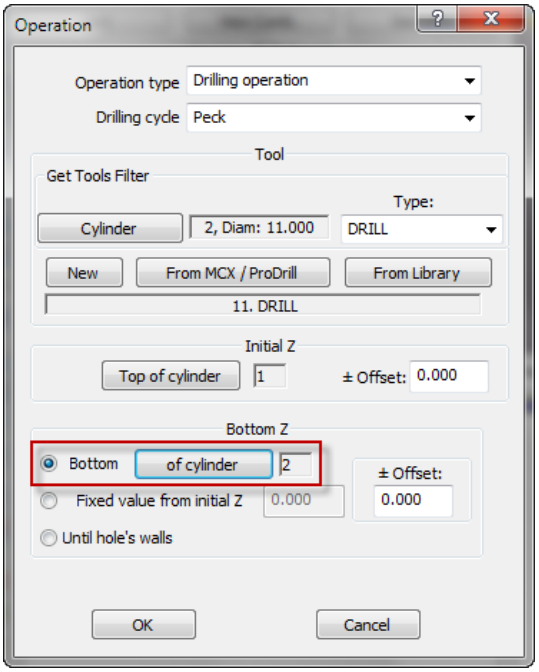
Define the next drilling operation as a **Peck** drilling cycle from the drop down list for drilling cycle. The diameter of the form to be drilled can be associated to the graphics image itself. Currently the Cylinder to drill is associated to the first cylinder - which is the chamfer cone. To change this, click on **Cylinder**. You can now highlight any cylinder on the graphics. Highlight the 11mm cylinder. Once you see the rectangle shape (as shown below) over the desired cylinder, it indicates that you have chosen the right one!



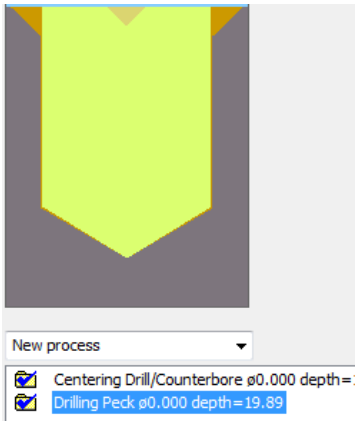


The result of the selection is as shown below. Cylinder now indicates **2. Diam: 11.000** - indicating the actual diameter of that cylinder**.** You need to select a 11mm drill now from the library. Choose a 11mm drill tool for this form using the "**From Library**" option. The following is the result of both settings. Once the tool is chosen, you need to set the depth for the form. Ensure that the "**Bottom**" radio button is checked and click on "**of cylinder**". Now click on the same cylinder from the graphics as above. This will set the bottom for the depth to be the bottom of cylinder 2.

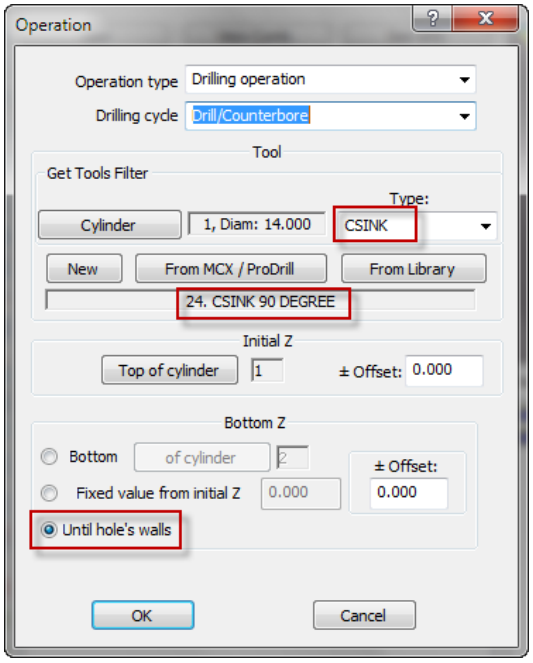


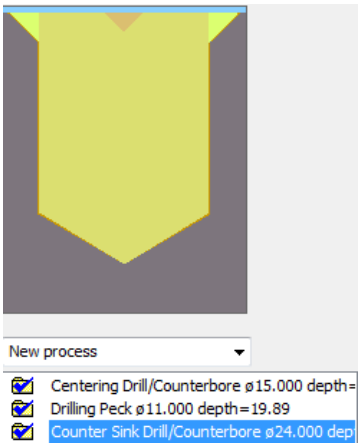


Click on **OK** to set the operation. When you click on the second drilling operation, you will now notice what happens with the second operation.

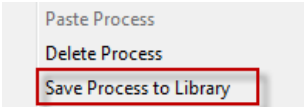


To define the final counter sink operation, right click and choose "**Append Operation**", select a tool type of **CSINK** and choose a **24mm counter sink 90 degree tool**. For the depth, since we are not sure about what the depth would be, you can use the 3rd option for the depth setting that states "**Until hole's wall**". *This setting will ensure that the counter sink tool is drilled to the maximum possible depth without violating any other cylinder or shape of the form!* Click on **OK** to accept the new operation. When you click on the 3rd operation, you will notice that the other 2 operations are also previewed before the counter sink operation. You will also notice that the counter sink operation does its job!





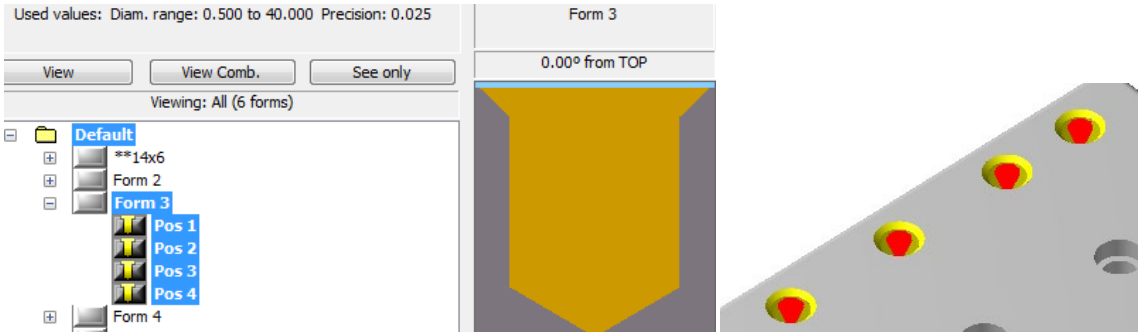
You have completed the process definition. To store this to the Library, right click and choose "**Save Process to Library**" The next time you encounter a similar shape and form, the same process will be applied to the form! To rename the process or modify the process, you can visit the Forms Library Manager dialog, click on the form and modify it.



MACHINING ONLY SELECTED POSITIONS BY CLONING FORMS

When ProDrill identifies a form, all the positions (XYZ coordinates) are listed together under the same form name - independent of the toolplane they are identified on. If you wish to machine only selected positions, you can do so by selecting the positions and creating a "Cloned form" with the positions listed in that group.

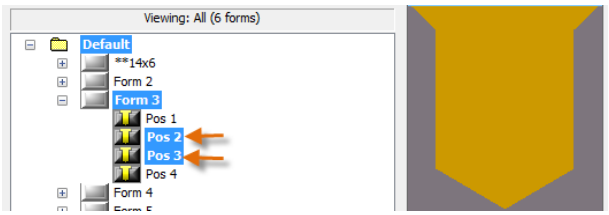
For the current part, let us review Form3 again.



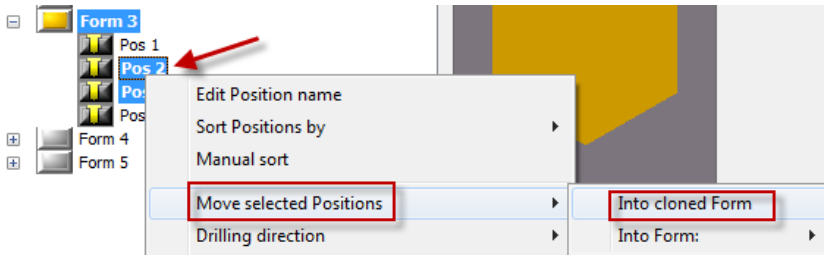
There are 4 positions for this form. If you wish to only machine 2 of the 4 positions, you can do one of the following:

OPTION1: SELECTION FROM THE TREE

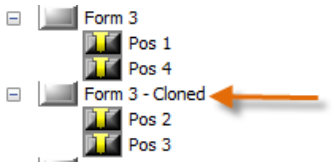
Highlight the positions from the tree and select them (by holding the CTRL key down the select more than one position).



We have selected **Pos2** and **Pos3**. Right click and choose the following option:



Note: Make sure the cursor is positioned close to the selected positions before right clicking. Click on "**Into cloned Form**". The 2 positions are now moved to a new Form group. The name of the group has the same name as the current form but with a suffix - **Cloned**. In this case, the name is "Form 3 - Cloned".

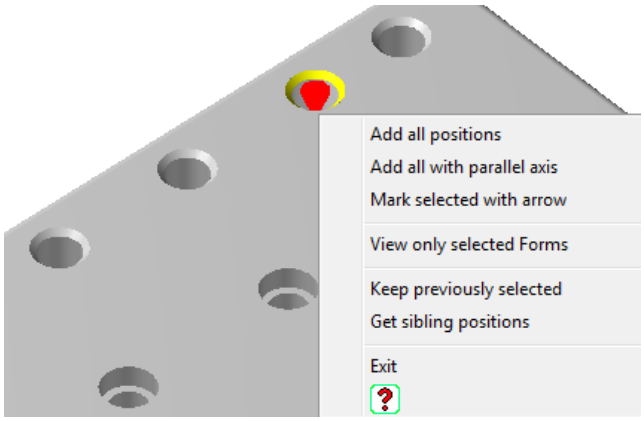


Now that these 2 positions are separated from the original group, you can choose to machine them using different machining processes if preferred OR to leave them alone.

OPTION2: SELECTION FROM THE GRAPHICS

ProDrill also offers intelligent tools to select forms from the Graphics area itself! This helps ensure that you choose the right positions before proceeding further. Note that when you highlight a drill form in the graphics area, the position and form are also highlighted in the tree!

Upon right clicking in the graphics area and highlighting a position, the following options are displayed:

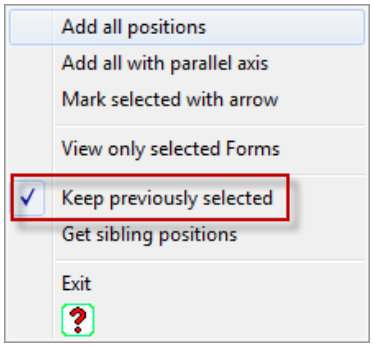


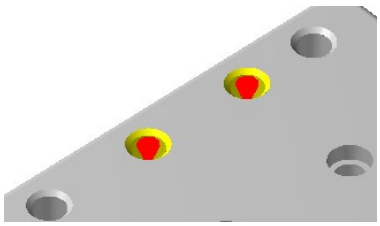
You can choose to add all the positions from the same form by selecting "*Add all positions*" or "*Add all with parallel axis*" to select all the forms found on the same construction plane.

When you highlight a position, it is selected. If you select if again or choose another position, it is unselected. There are options to keep the previous selection in this menu.

In our case, since we only want to select 2 positions, you need to select this option first to ensure that there is a check mark placed next to "**Keep previously selected**". When this is checked, you can continue selecting new positions without unselecting old positions.

Click on the 2 forms that you wish to select. The following is the result:





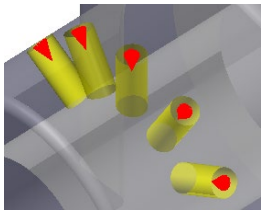
Now that the positions are selected, you can go through the same steps as described above to create the cloned forms.

You can also select different positions from different forms at the same time and create multiple cloned forms without having to split them form by form!

SORTING FORMS IDENTIFIED ON INCLINED PLANES

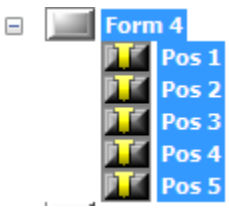
When forms are identified and positions listed in the tree, the ordering is not the desired method for machining the forms. This version allows you to re-order the positions to the desired preference for machining.

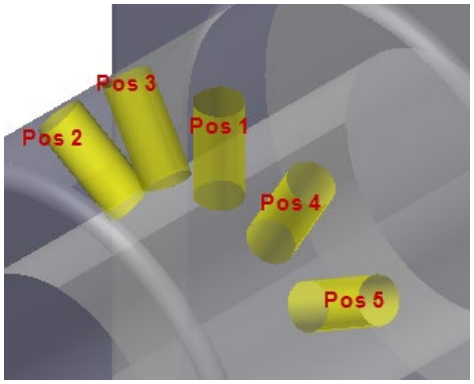
Note that ProDrill automatically creates new construction planes for the identified forms and names them in Mastercam WCS Manager according to the angle rotations relative to the plane that they are identified on!



PRODRILL 1 [30.00º FROM LEFT SIDE]
PRODRILL 2 [30.00º FROM TOP]
PRODRILL 3 [30.00º FROM TOP]
PRODRILL 4 [30.00º FROM RIGHT SIDE]

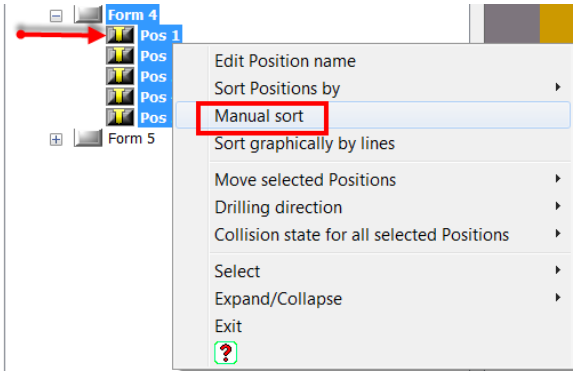
Click on **Form 4** and expand the positions. If you highlight each form, you will notice that the ordering for the drilling is not optimal for machining. Each position is also identified on a different construction plane.





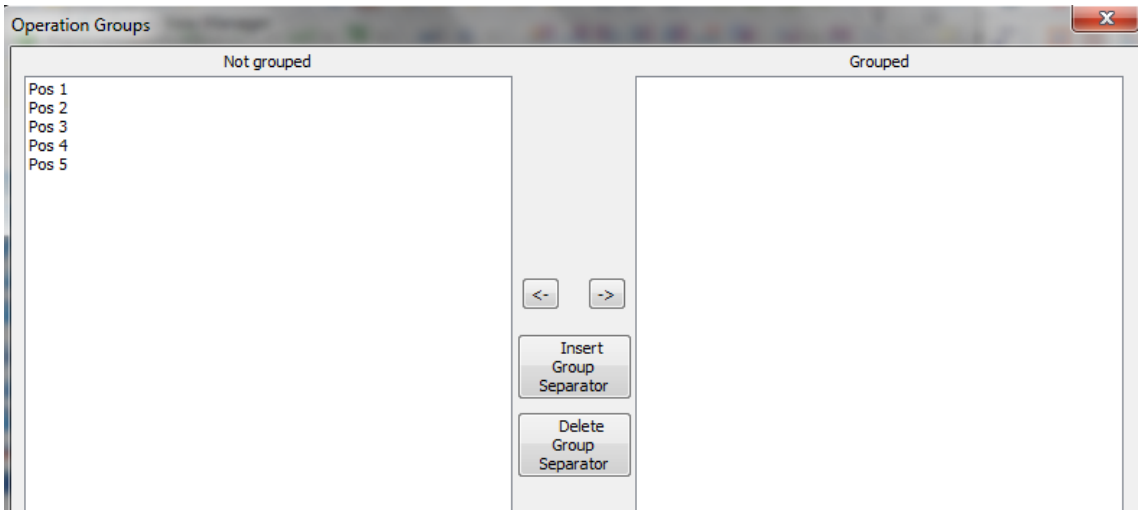
OPTION1: SORTING POSITIONS USING MANUAL SORT METHOD

To sort these forms in a preferred order, you can do the following: Right click on Pos1 as long as all the positions are highlighted and select **Manual Sort**.

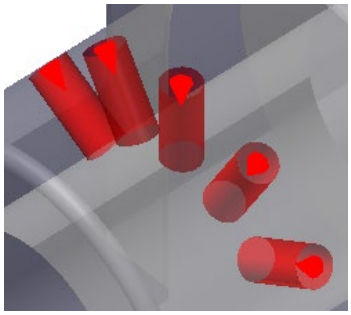


Note: The selection method is important. If you click on Form 4, all the forms will be added to the Layout Dialog. ***You have to ensure that you right click when the cursor is on top of Pos 1!***

The following dialog is presented with the 5 positions listed in the dialog.

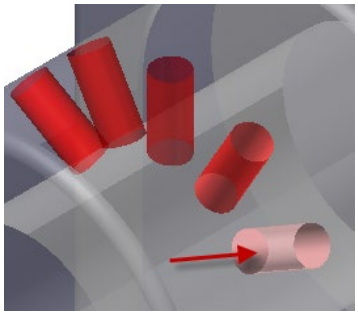


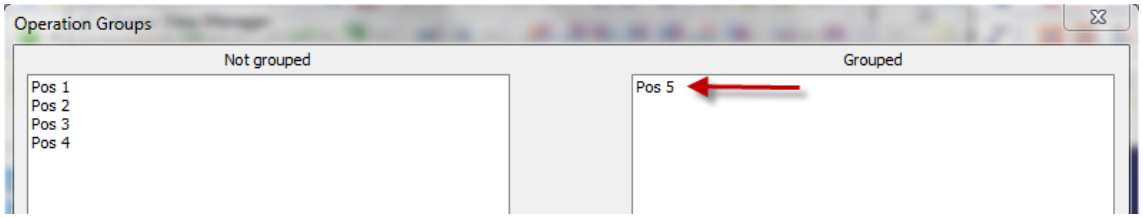
The graphics area also has the 5 positions highlighted in red.

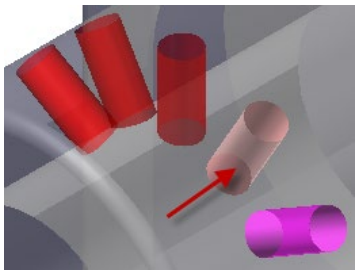


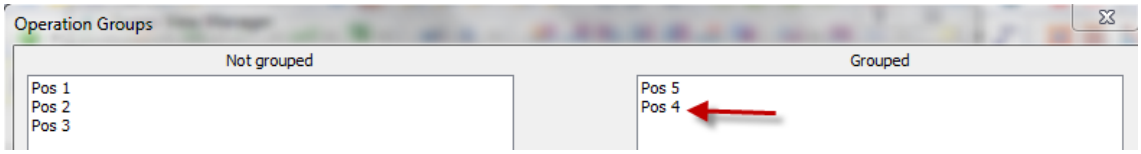
To sort the positions, you need to move the different positions in the desired order to the right side of the layout dialog.

The best method to do this is graphically. Click on the red cylinder from the graphics are that you wish to have first on the list. That position is automatically moved to the right side. Each position that is moved is highlighted in purple color.

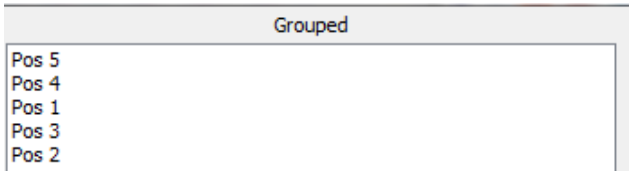






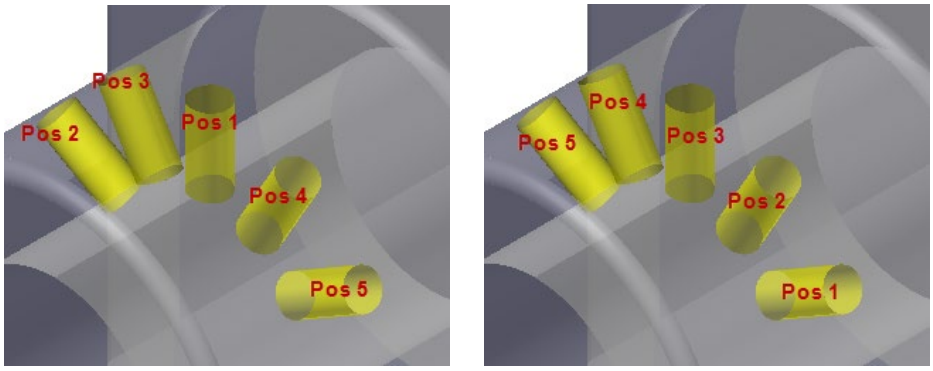


You can move the rest of the positions to the right side in the same manner.



The result of sorting the positions is as follows:

BEFORE **AFTER**



The tree list also displays the new change:

Now when toolpaths are created, the order is respected.

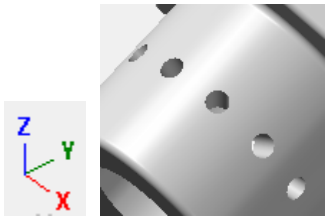
NOTE: You can **also move the positions** manually up and down the list by:

1. **holding** the **ALT-key** down
2. **highlighting** a **position** to move
3. using the **up** and **down arrow key** to move the position to the new location!

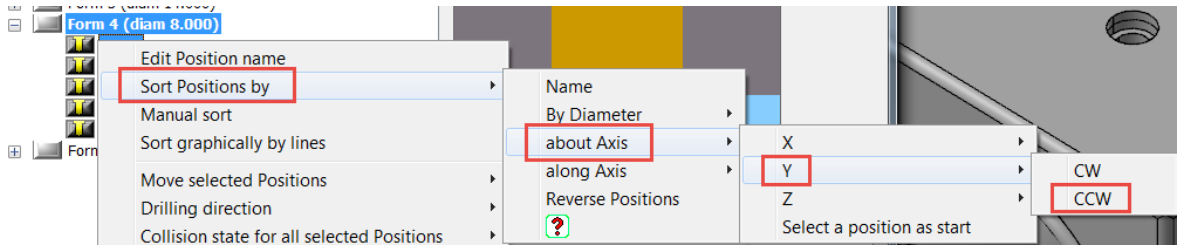
OPTION2: SORTING POSITIONS USING AXIS SORTING METHODS

ProDrill offers different sorting methods for forms and positions found on different planes.

Along Axis and **About Axis** are two very powerful sorting tools available. For this part, we see that the positions can be sorted about the Y axis in a clockwise or counterclockwise direction.



Highlight the 5 positions as above, right click and choose the following option:



The positions are sorted about the Y axis in a counter clockwise direction and the result is as follows:

Before: After:

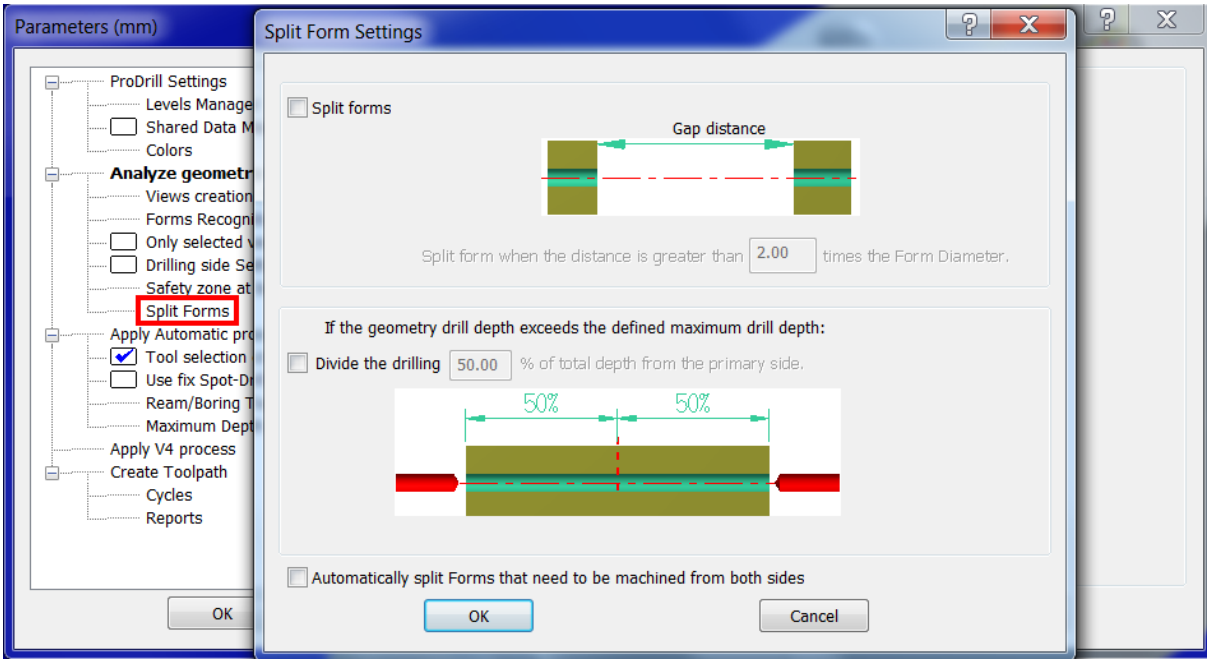
You are now ready to machine these forms in the order sorted.

You can also sort positions on the same plane using this method or the manual sort method.

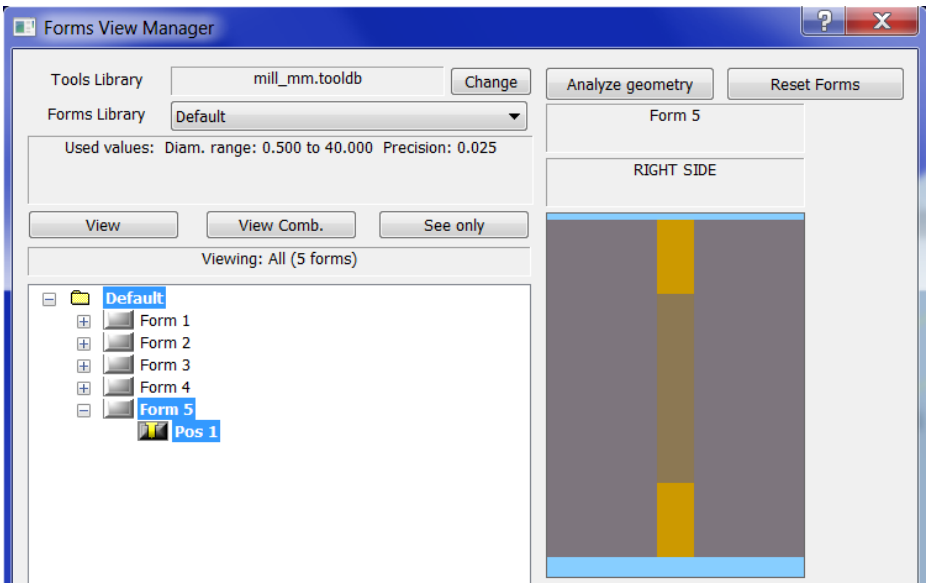
SPLIT FORMS TO BE DRILLED FROM OPPOSITE PLANES

ProDrill V5 allows you to split the forms to be machined from opposite sides from the Forms View dialog itself.

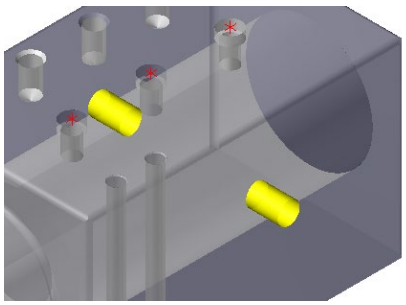
Before doing this, visit the **Settings** tabs and ensure the Setting for splitting forms of this nature is disabled:



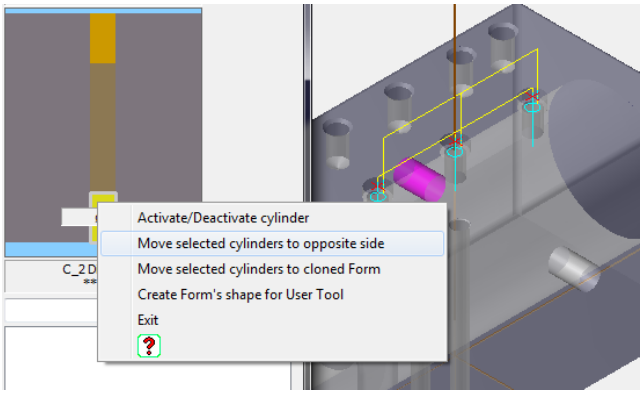
Re-process the part by clicking on **Reset Forms** and **Analyze Geometry**. Click on **Form 5**. The following form is displayed in the dialog.



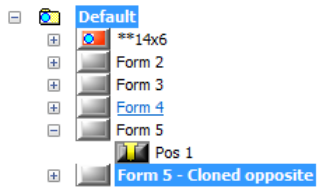
The graphics highlights the form also. You can clearly see that the form has a large intermediate space in between.



To split this form and machine it from opposite sides, you can click on the appropriate side that you wish to move to the opposite side, **right click** and choose **Move selected cylinders to opposite side** from the graphics display area.

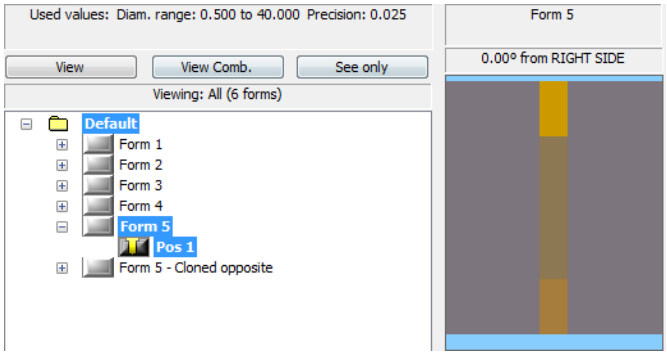


The form is now split into 2 and a new cloned form is created in the tree.

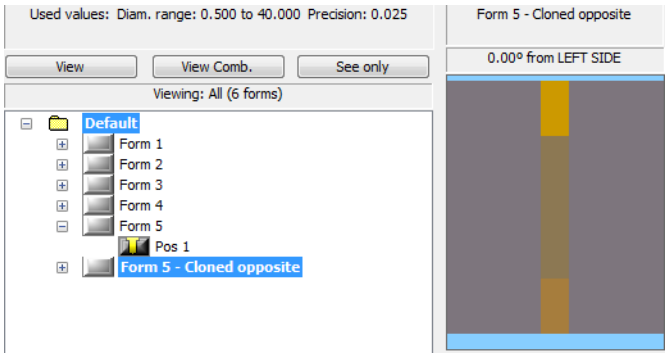


Highlighting that form will display that section in the graphics area with the opposite side grayed.

Main side



Opposite side



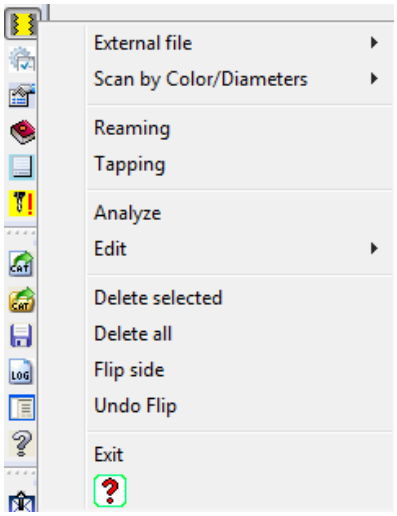
Note that the opposite side is now identified in the form name also. ***You can choose multiple cylinders to move to one side or the other!***

SETTING TAPPED AND REAMED FEATURES

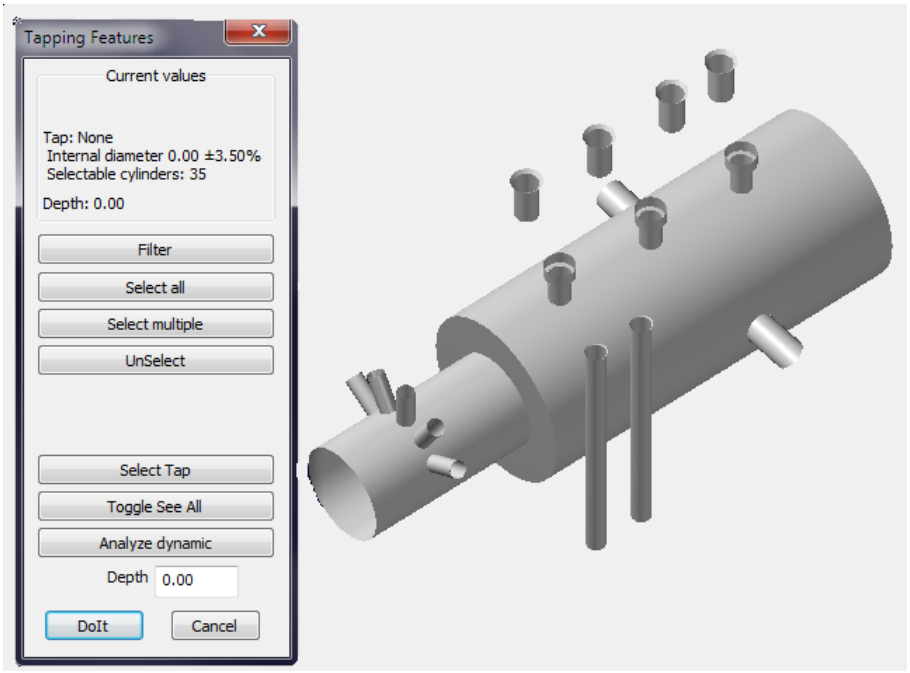
You can assign forms to be tapped or reamed using ProDrill's Set Features option. When forms are tagged in this manner, the drilling processes will automatically add a tapped or reamed toolpath operation.

SETTTING TAPPED FEATURES

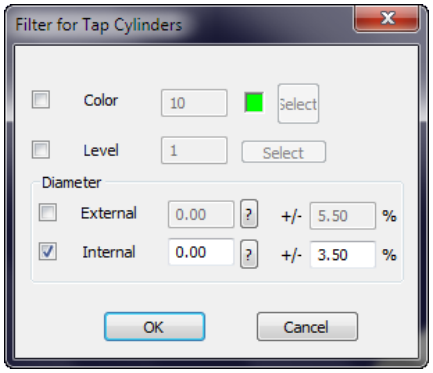
Click on the **Set Features** icon. The following options are presented:

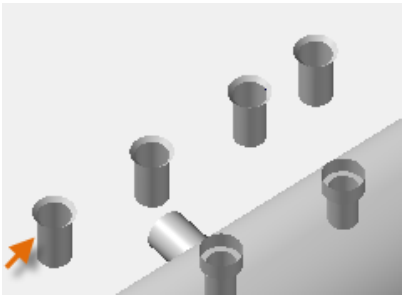


Click on "**Tapping**". The cylinders found in the part are presented along with the dialog to select the positions.

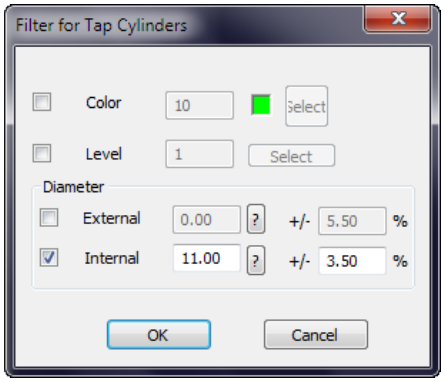


Click on the **Filter** option from the menu. The following dialog is displayed:

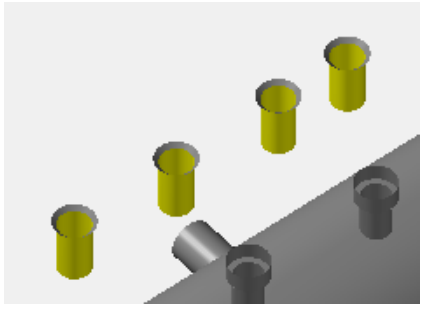




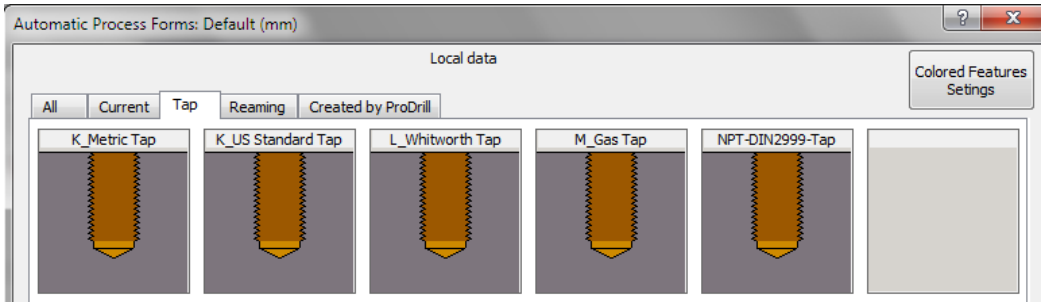
We can isolate cylinders using different filtering techniques using this dialog. To isolate the forms as shown to the right, click on ? and click on the indicated cylinder. The filter dialog is now updated to the value of the cylinder:



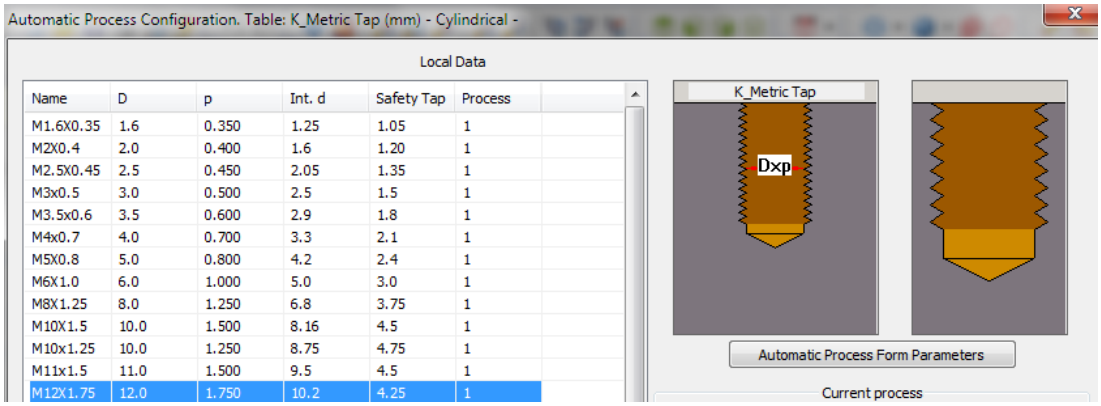
The cylinders that match the filtering options are highlighted in the graphics area. The menu also indicates the number of entities that match the filtered criteria. From the menu, click on "**Select All**". All 4 positions are selected and highlighted:



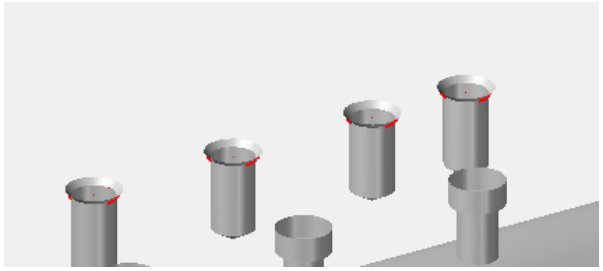
Click on **Do it**. The following dialog is presented with the different tap form libraries available.



Select the "**K_Metric Tap**" library and choose the **M12x1.75** form.



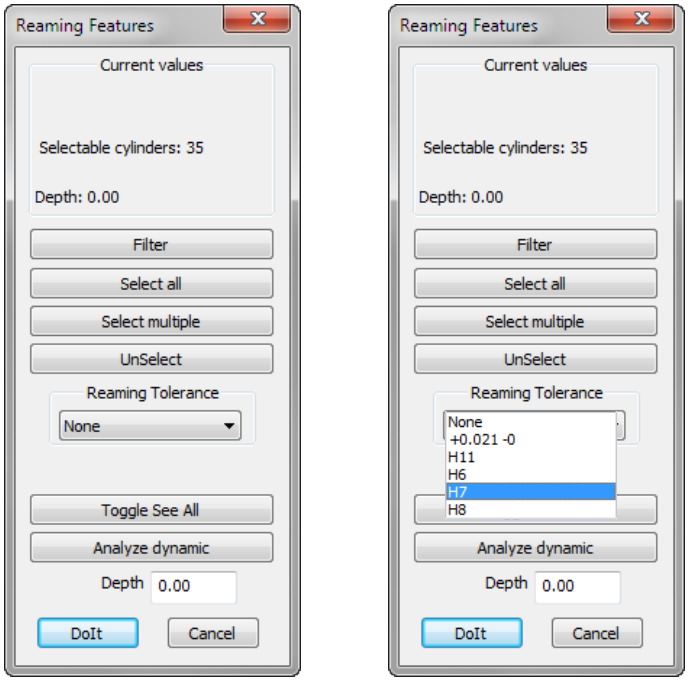
Click on **OK** to accept the selection. The 4 cylinders are marked for tapping. The forms are identified by 270 degree red arcs created around the cylinder.



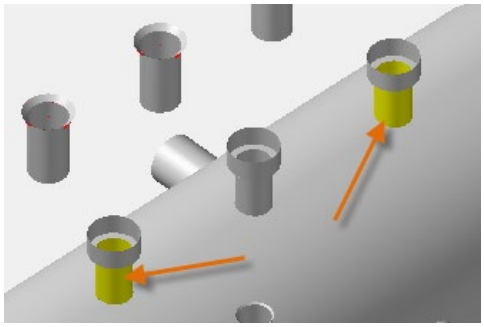
Now the forms can be processed using the Forms View Manager.

SETTTING REAMED FEATURES

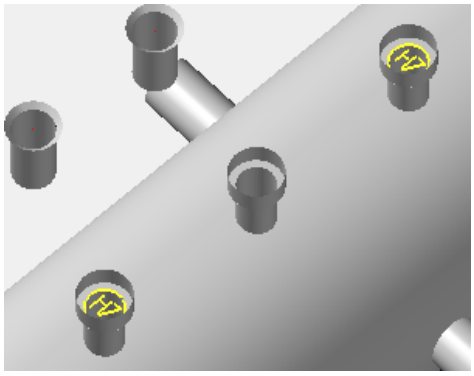
Reaming tolerances can be set in the same manner as the tapped forms. From the Set Features menu, click on **Reaming**. We can choose to set the forms as **H7** tolerances. Choose the H7 from the drop down list as shown:



We will set the lower cylinders of 2 counter bored forms as H7 reamed holes. Click on the following cylinders. They will be highlighted as shown:

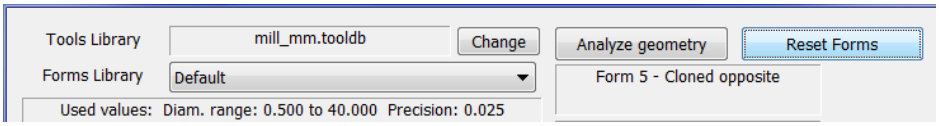


Click on **Do it** to set the forms as reamed forms. The reamed cylinders are identified by 360 degree yellow arcs at the top of the corresponding cylinder that needs to be reamed. The characters H7 are also written on top of the cylinder.



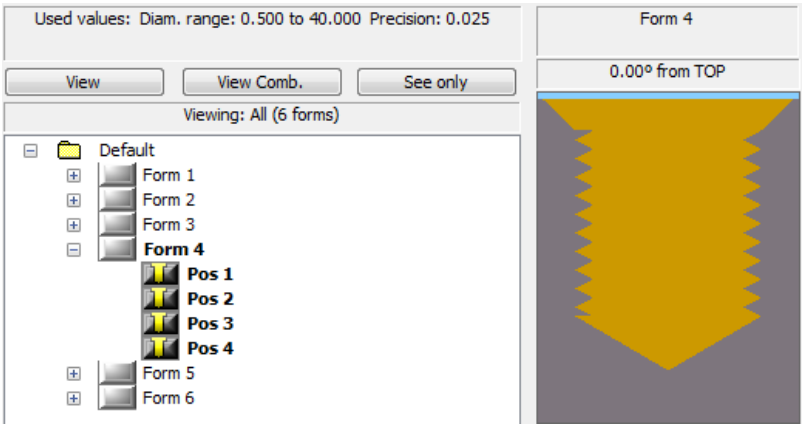
You are now ready to process the part that was identified for reaming and tapping.

Launch the Forms View Manager. Click on **Reset Forms** to remove the previous work that was done on the part. The tree is now clean.

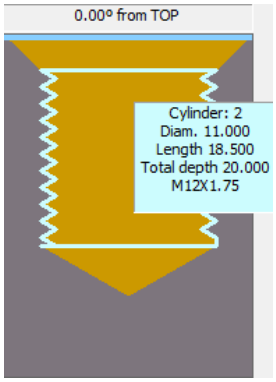


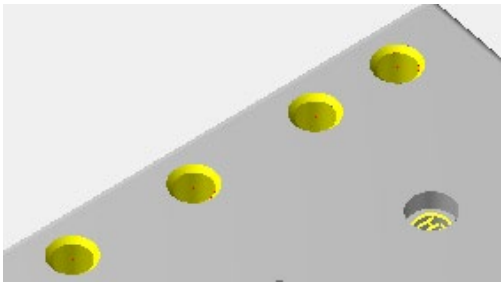
Reprocess the part by clicking on "**Analyze Geometry**". The forms are analyzed again and ProDrill now identifies the tapped and reamed forms and displays them with graphical identifiers.

The tapped forms (Form 4) are grouped and displayed as follows:



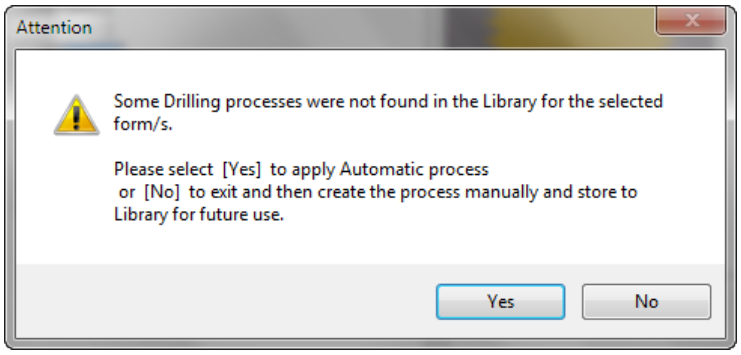
Upon placing the cursor above the tapped cylinder, the following information is displayed:



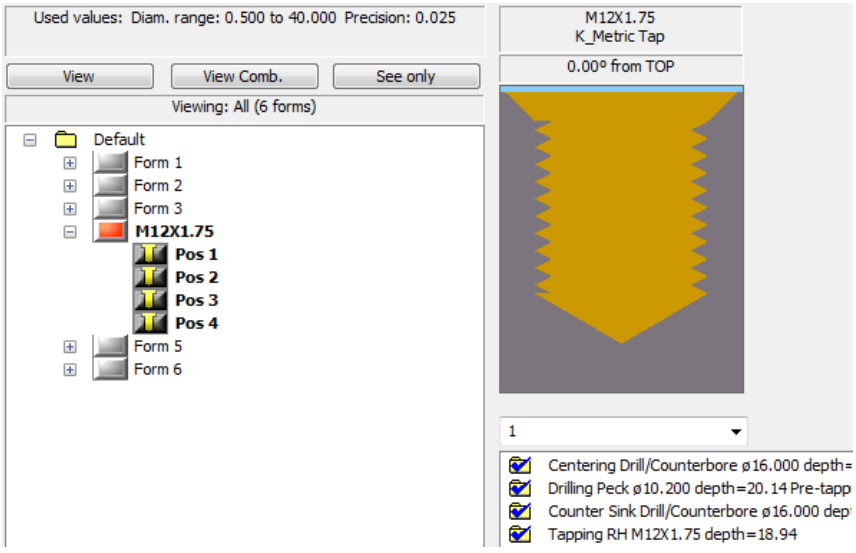


Apply processes to them by right clicking and choosing "**Apply from Library**"

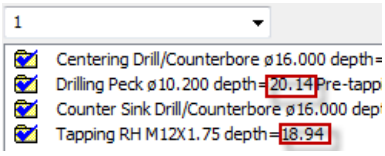
Since there is no process in the library, the user is prompted to create an automatic process:

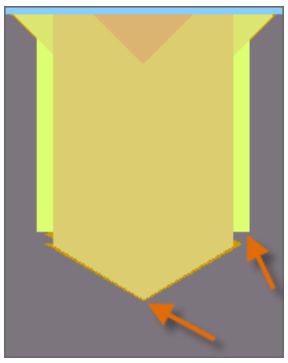


Click on **Yes** from the above dialog. The following processes are applied and the form name and description are also updated:



A *safety depth for the tapping cycle is automatically applied from the tap library*. Notice the drilling depth is calculated to 20.14 mm and the tapping depth to a lower value.

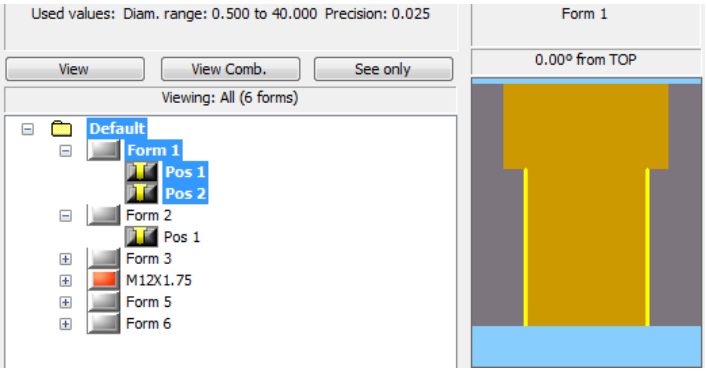


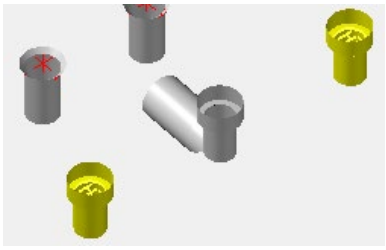


Upon doing a graphical tool use (by clicking on the different drill cycles), you notice the above - the tap tool stops at a safety distance.

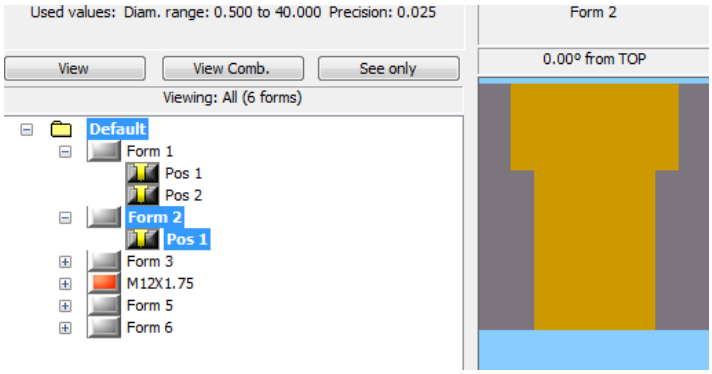
If you notice this analysis, there are 6 forms.

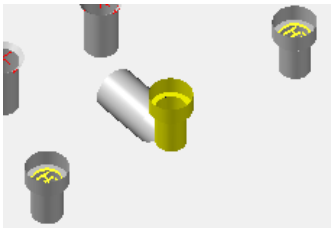
If you click on **Form 1**, you will notice the 2 reamed forms identified with yellow lines along the cylinder marked to be reamed. The corresponding cylinders are also highlighted.



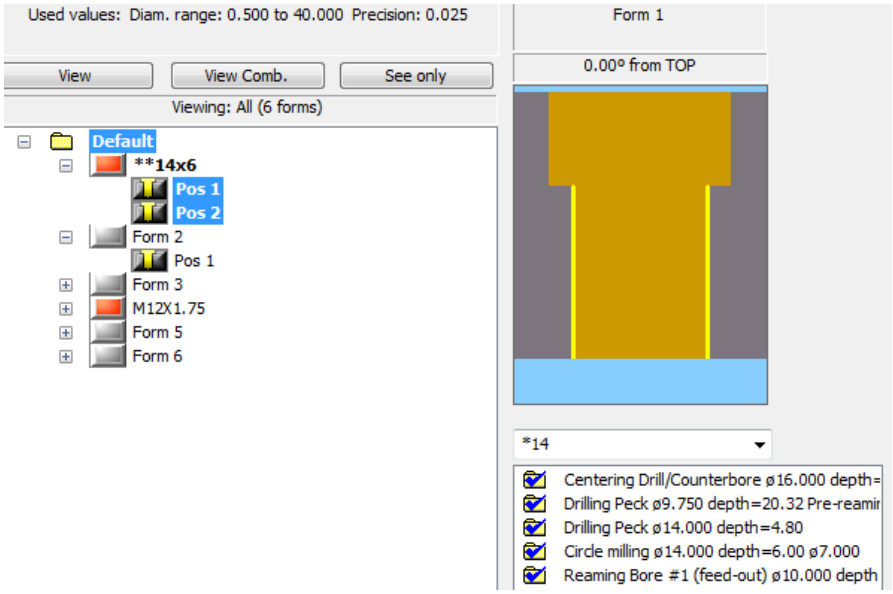


If you click on **Form 2**, it is the same counter bored form with the exception that it wasn't marked for reaming.





Add drilling processes by " **Apply Library (if it fails do Automatic)"** to "**Form 1**". A ream cycle is added automatically to the process.



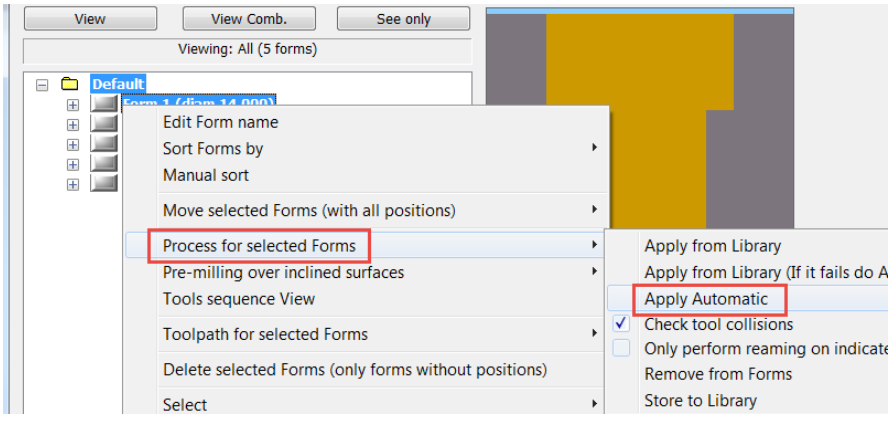
Safety clearance depth is also added to the ream cycle from the A_Drill table.

CREATING MULTIPLE PROCESSES, EDITING AND STORING TO LIBRARY

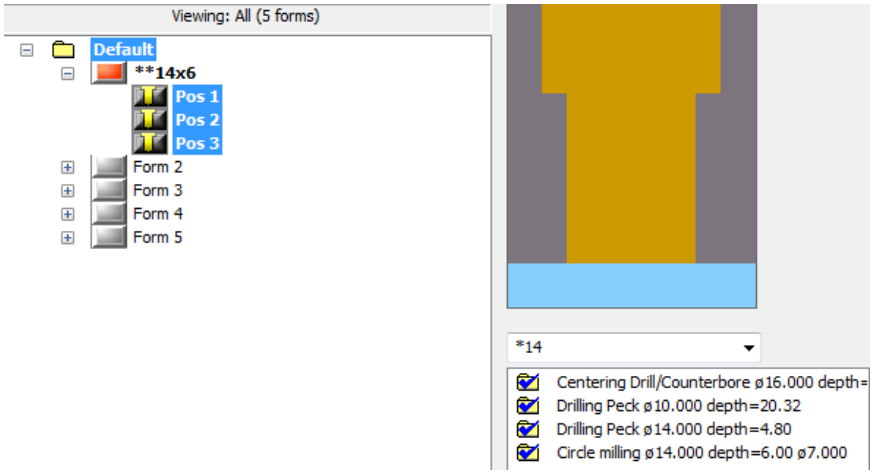
Reload **PART_4.MCAM**. This will clear any reamed or tapped features added.

Analyze the part again by clicking on the **Forms View Manager** icon. We will now create multiple processes for the 3 counter bored holes.

Apply "**Automatic Processes**" to the counter bored forms.



The following processes are applied to the 3 positions.

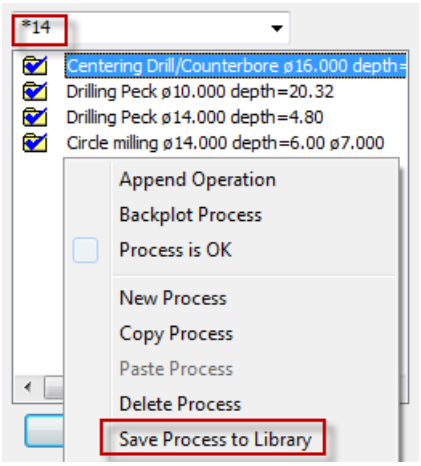


Lets apply a different process to Position 2 (Pos 2). In order to apply a separate process, lets create a cloned form for Pos 2.

Now that it is separate, we can start creating multiple processes to machine similar forms.

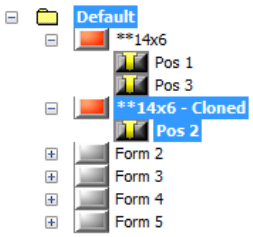
Before creating a new process using the existing one, let us first store the newly added process to the V5 Library.

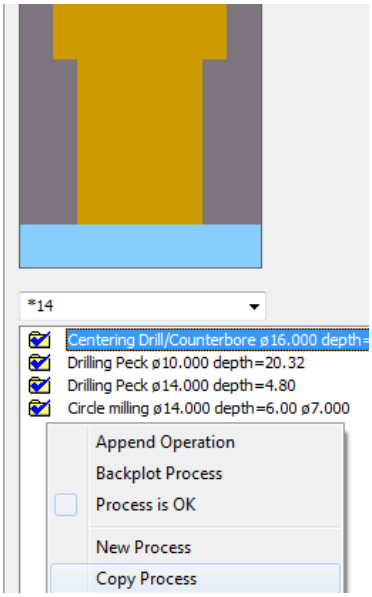
To store this to the library, right click from the Process dialog and choose "**Store Process to Library**".



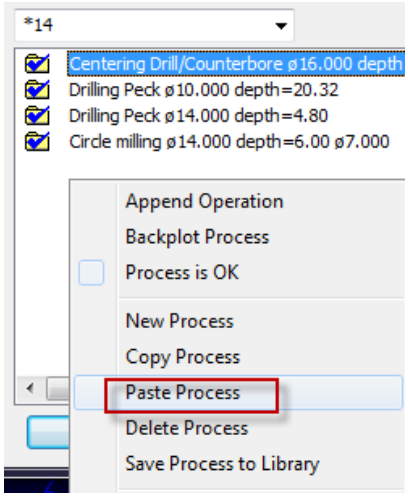
The process is now stored to the V5 library under the form name **14x6 and the process name as **"*14**". Next time you process a similar part, when you choose "Apply from Library", the process is applied from the V5 Library.

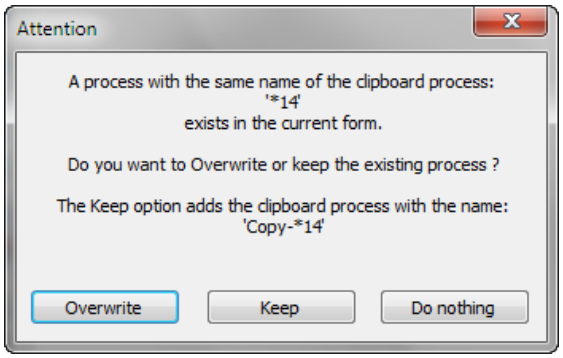
To make a copy of the same process and store under a new name, **right click** and select "**Copy Process**".



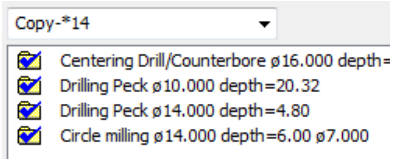


Once you have made the copy, right click again and choose "**Paste Process**". The following warning message is displayed indicating that you are attempting to paste a similar process and automatically recommends a new process name.



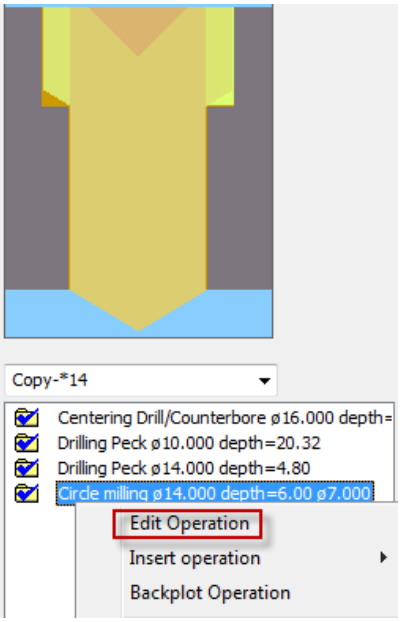


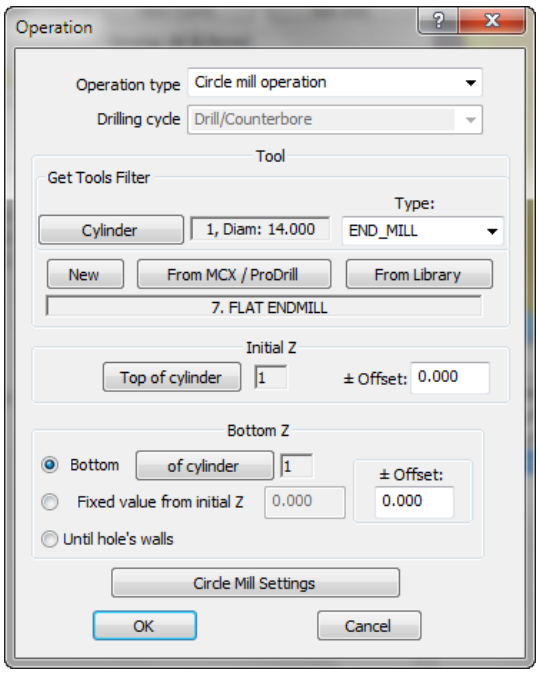
Click on "**Keep**" to make a copy. The process is now copied and named as "**Copy-*14**".



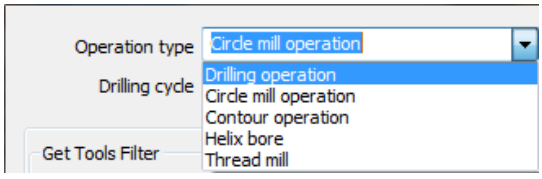
Let us now change the process for the circle milling operation to a drilling operation with an end mill.

Right click on the "**Circle milling**" operation and select "**Edit Operation**". The following dialog is presented:

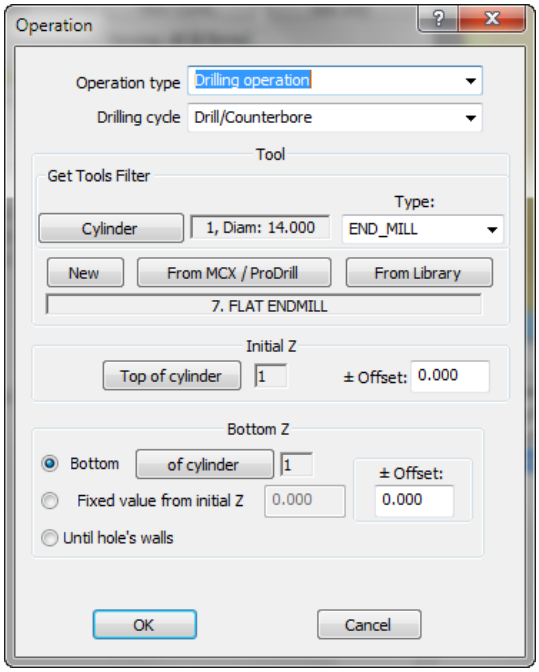




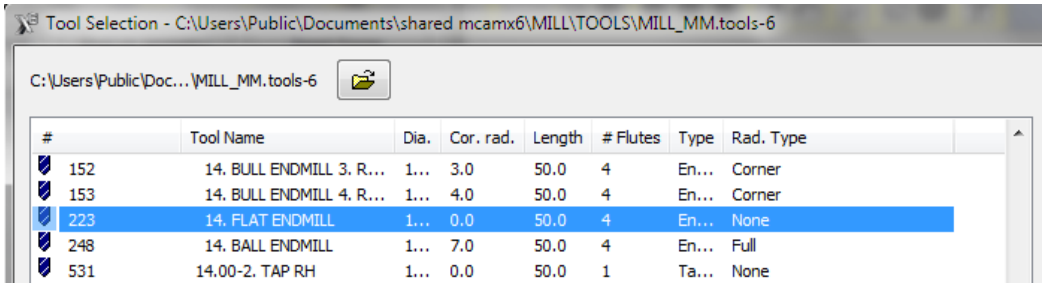
Pull down on the "**Operation Type**" and choose **Drilling operation**.



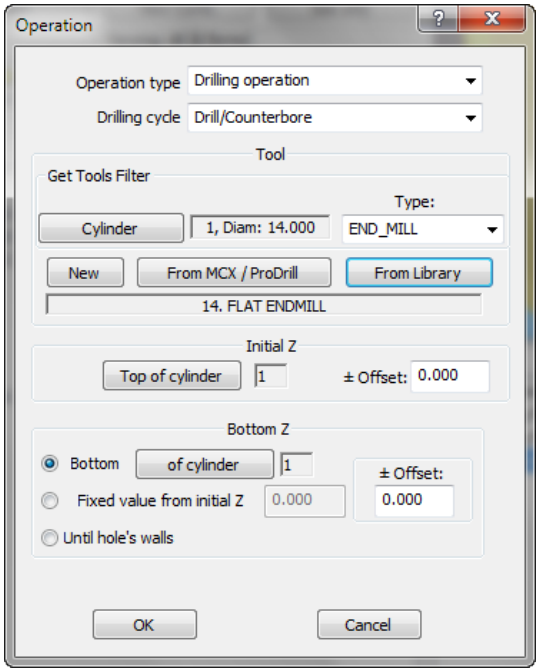
The dialog changes to the appropriate settings. The only other change that needs to be made is to the tool. The current tool selected from the library is the 7mm end mill (tool to perform circle milling operation.



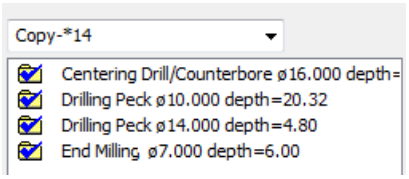
To select a tool from the tool library, click on the "**From Library**" button. Choose a **14mm flat end mill** tool.



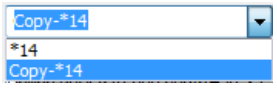
Now that the right tool has been selected, the dialog is now set for drilling with an end mill.



Click on **OK** to close the dialog. The following is the result for the list of operations. Note that the process name "**Copy-*14**" now has a drilling cycle instead of an end milling cycle.



The drop down menu displays the 2 processes available.



If you select Form 1, you will notice that the Circle milling operation is still listed.

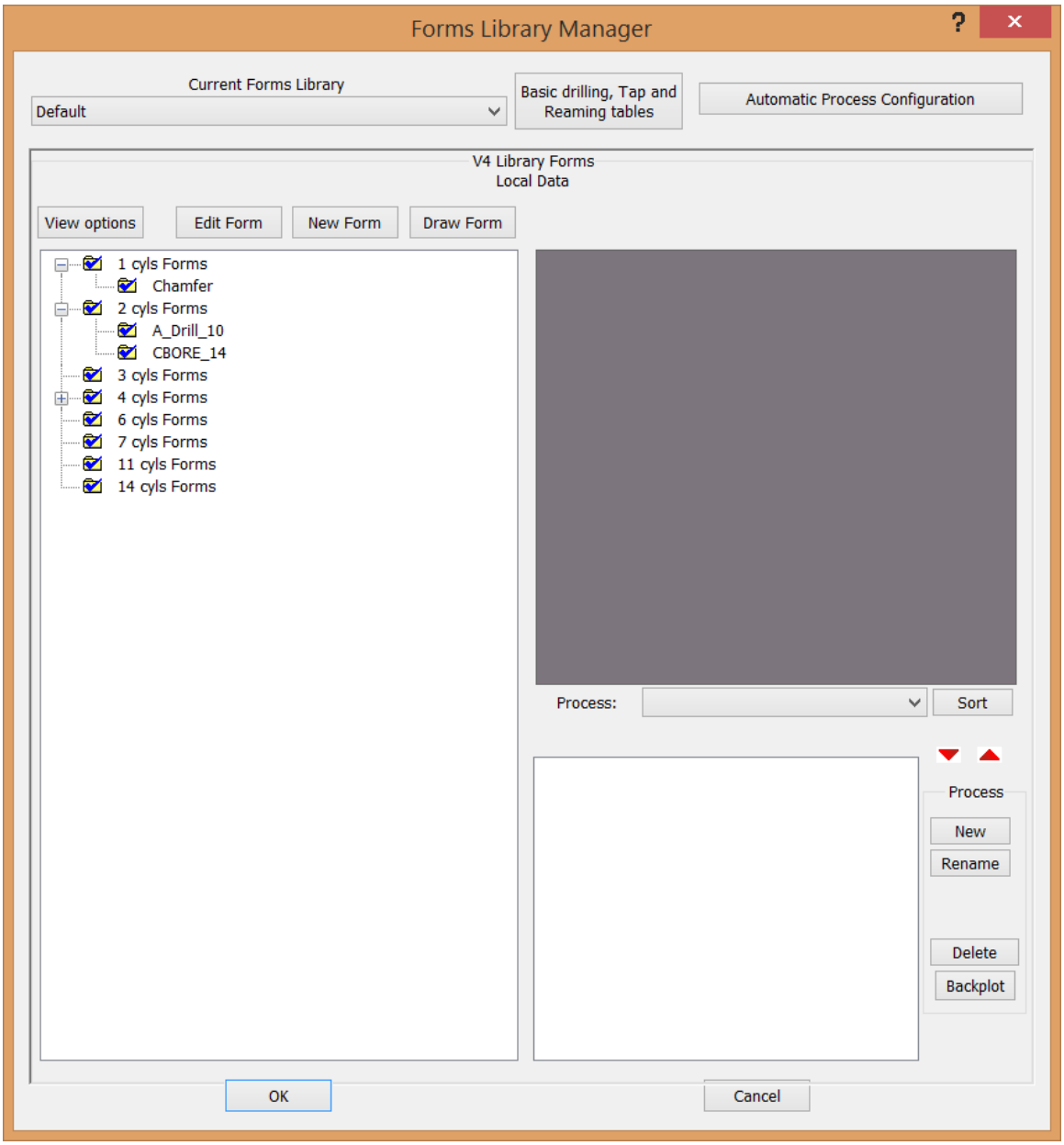
You have now split the form by cloning one of the positions, created a copy of a process, edited it and applied a different process to the same form type.

V5 LIBRARY

To review the processes stored to the library, click on the Forms Manager icon.

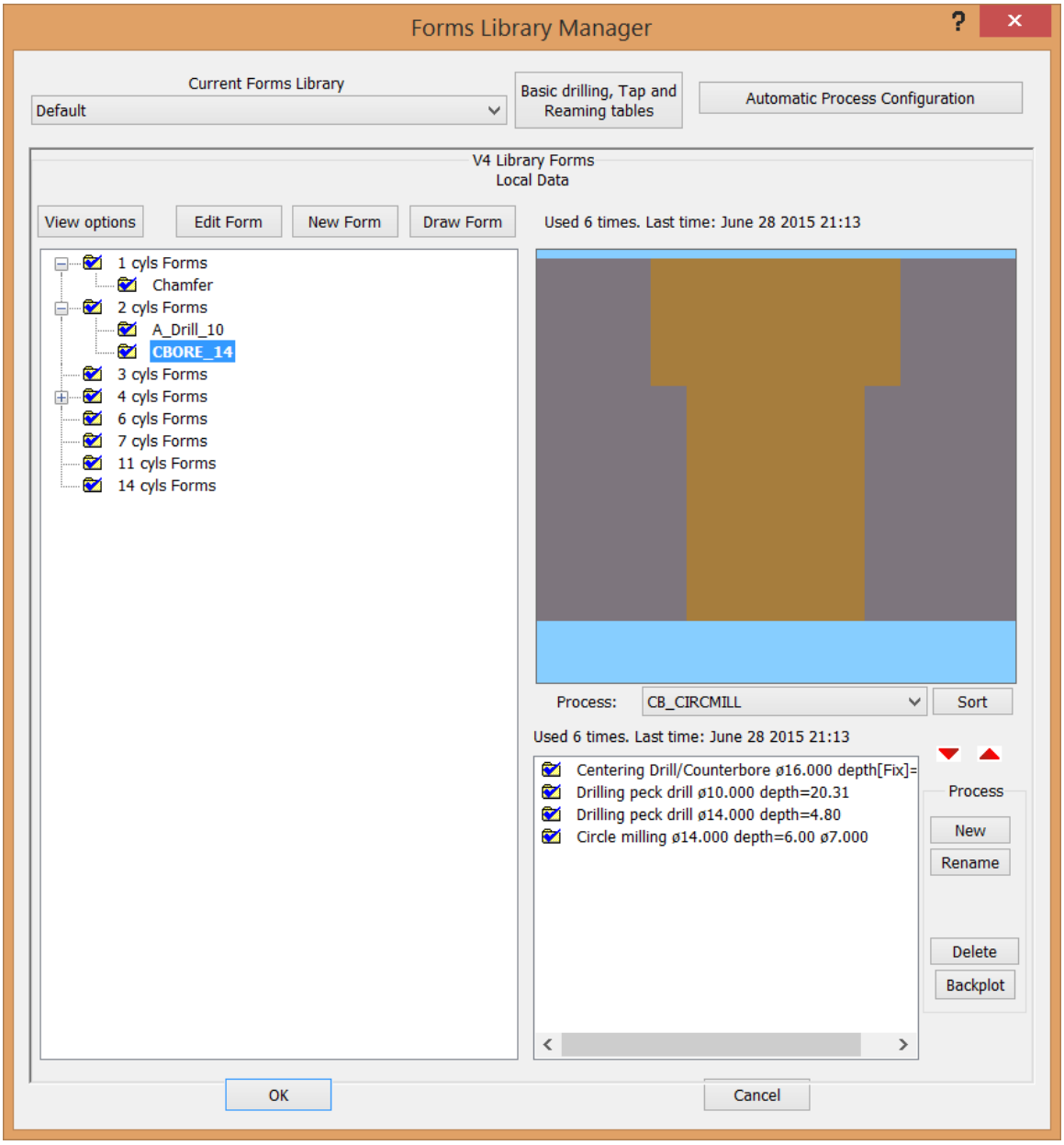
The Library Manager dialog is opened. The forms are listed here according to the View options settings. Here they are grouped by number of cylinders and forms. If you have stored more forms or processes to the library, they are listed here.

You can now edit, review or create new forms from this dialog.

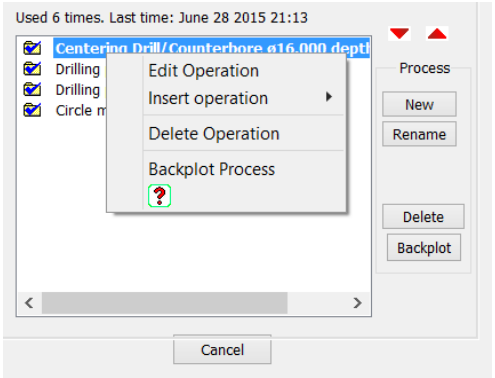


Click on the form **C_BORE14.**

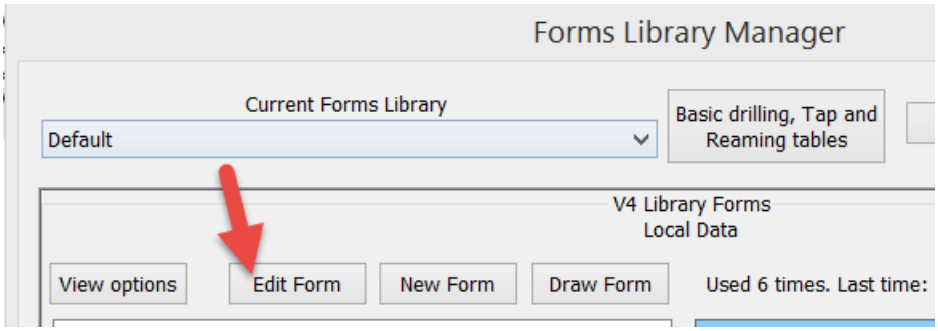
You will see the form shape displayed and all the options to manipulate it.



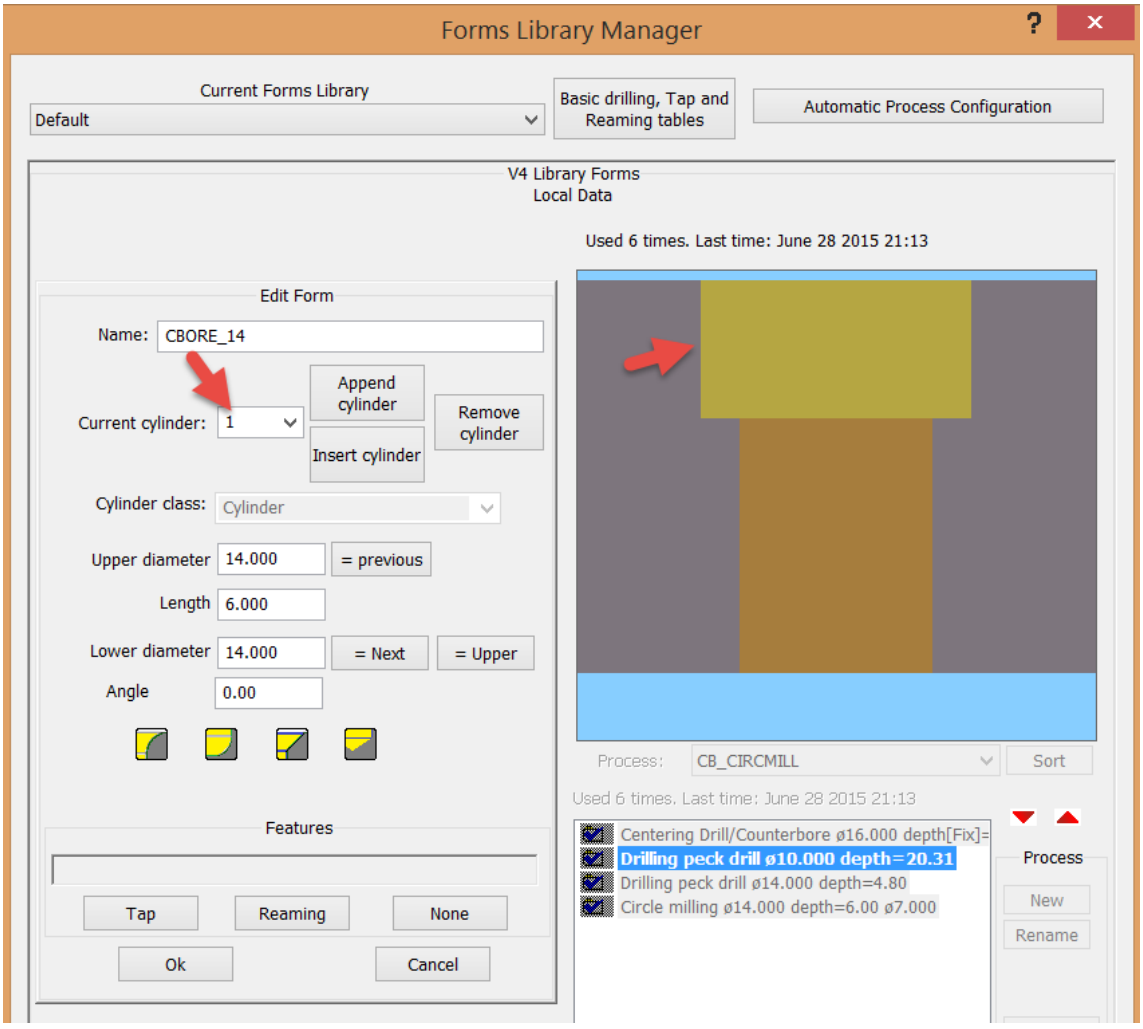
You can right click on any of the operations and edit them using the following options if desired:



If you click on the **Edit Form** button from the top:



You can edit either any of the form's cylinders, modify the size, length, etc. and add additional cylinders if needed. You can also set any of the cylinders aa reamed or tapped holes.



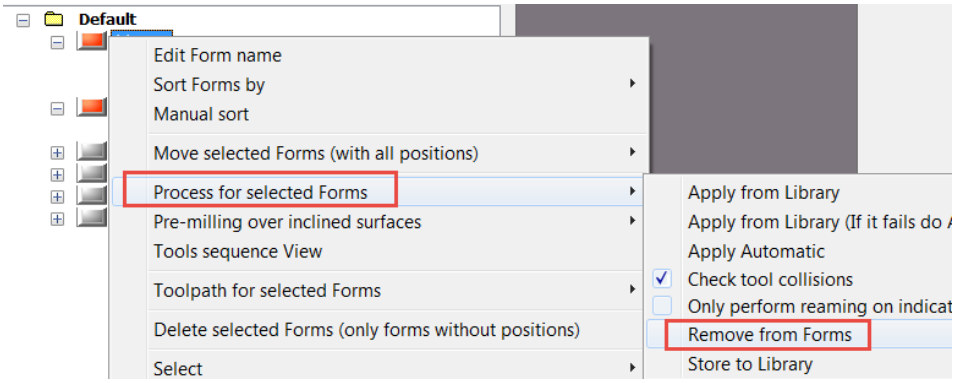
If any changes are made, the library is automatically updated and a message is displayed that the form was updated.

You can also create new forms from this dialog and draw the forms as surfaces or Solids.

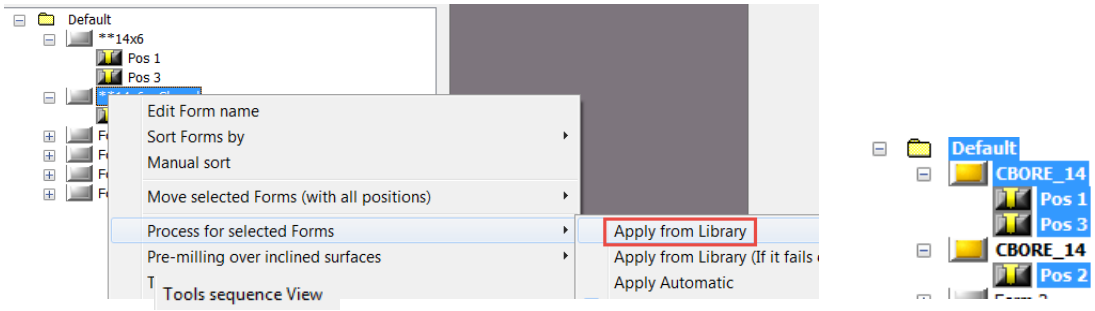
APPLYING PROCESSES FROM THE LIBRARY

Now that you have stored some processes to the V5 Library, let us apply these processes to the part again.

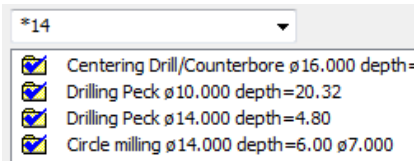
Launch the Forms View Manager and remove the processes from both the forms.



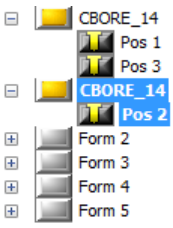
Click on "**Apply from Library**" for both the counter bored forms. You will see that the processes are applied from the library and the icon is orange for the form folder. The name of the process is the same for both processes.

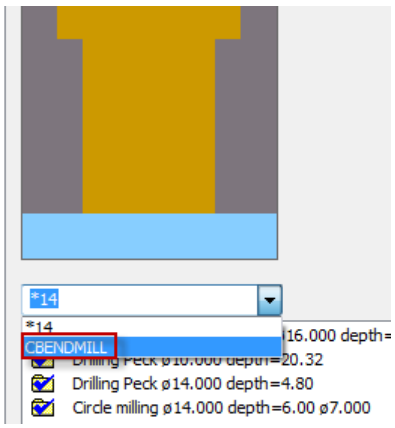


If you review the process list, the process is also the same for both forms.



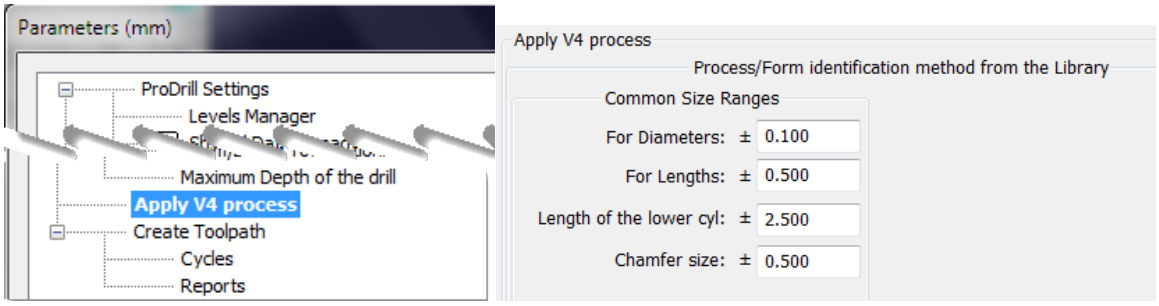
You can click on the drop down box and choose the other process available. Choose the **CB_ENDMILL** for the second form.





The above exercise shows how easily you can define and use multiple processes to be used with a similar form type.

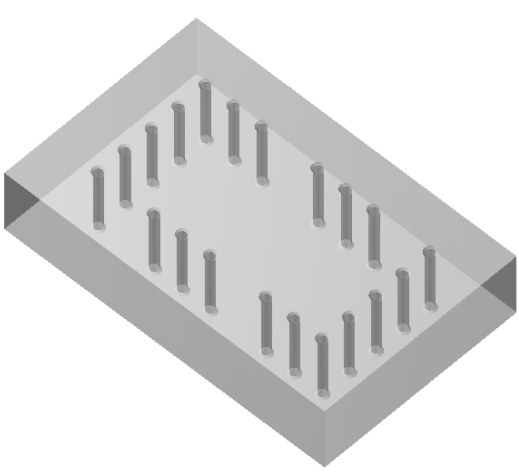
Note that the following **Setting** controls how Library forms are applied to new forms identified.

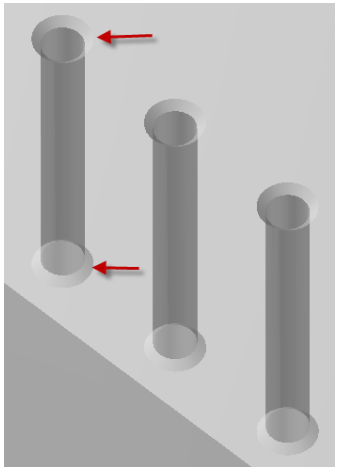


Setting a large value for the length of the lower cylinder can allow for most forms of similar types but varying minor diameter length to use the same process. If the value is small, ProDrill will prompt you with processes that match the form identified!

SPLIT CHAMFERS ON OPPOSITE PLANES AND MACHINE

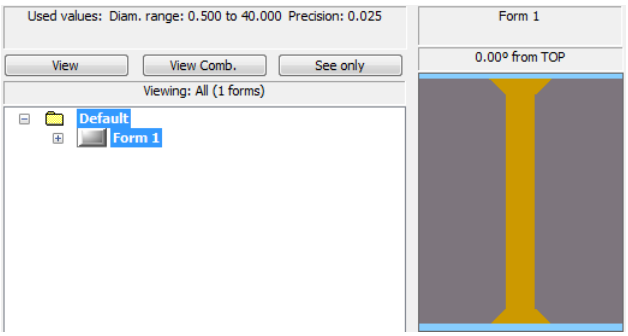
Load the part: **SPLIT_CHAMFERS_ON_OPPOSITE_PLANES.MCAM.** The part has drill forms that have chamfers on opposite planes.



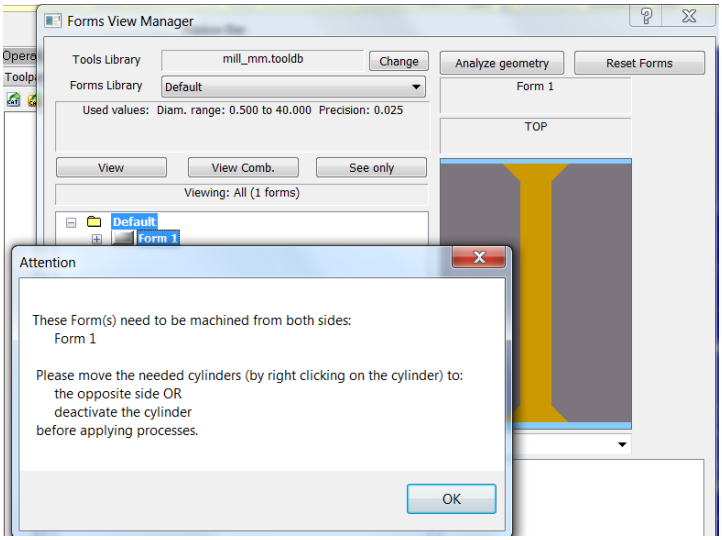


Attempting to machine these forms without easily splitting one of the chamfers to the opposite plane can be a time consuming task. ProDrill offers some intelligent strategies to machine such forms.

Process the part by launching the **Forms View Manager**. The forms are identified with chamfers on both ends.

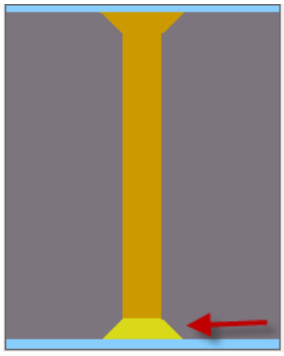


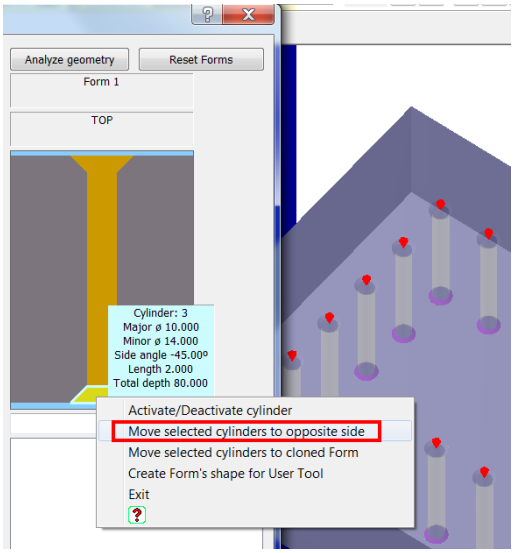
If you attempt to apply processes to this form, you are warned that these forms need to be split prior to applying a process.



SPLITTING THE CHAMFERS TO OPPOSITE SIDE

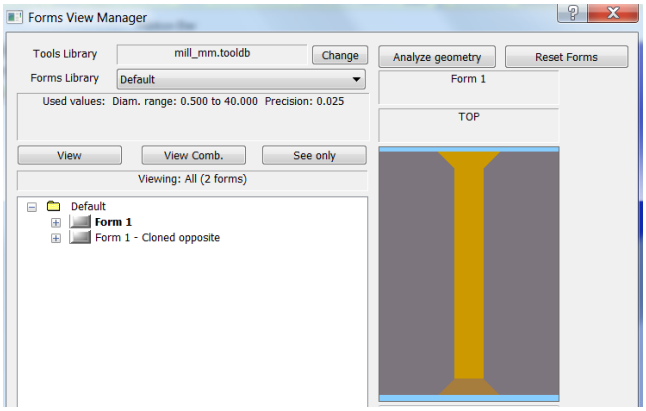
In order to machine this properly, we will need to apply the proper drilling processes from one side and then apply the chamfering process from the other side. To facilitate this process, you can highlight the bottom chamfer, right click and choose **Move selected cylinders to opposite side**.

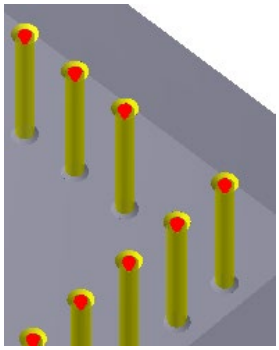




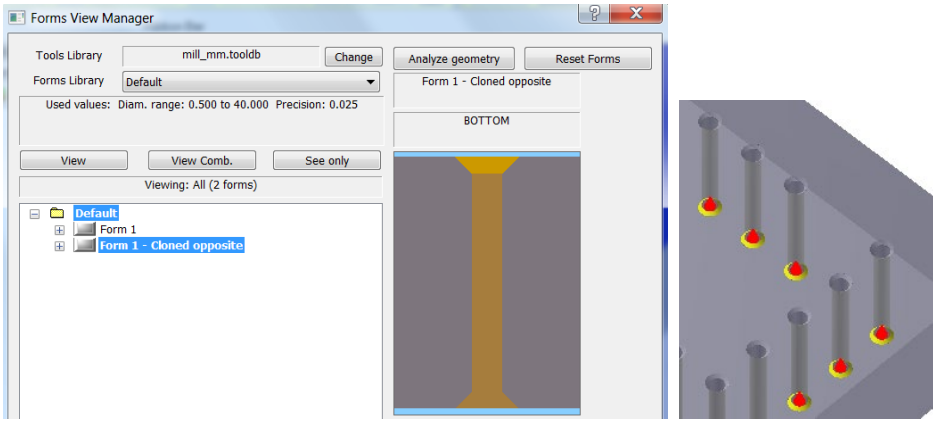
All the chamfers from the form are now moved to the opposite side. Note that ProDrill will not modify the actual geometry of the part in the process.

TOP SIDE



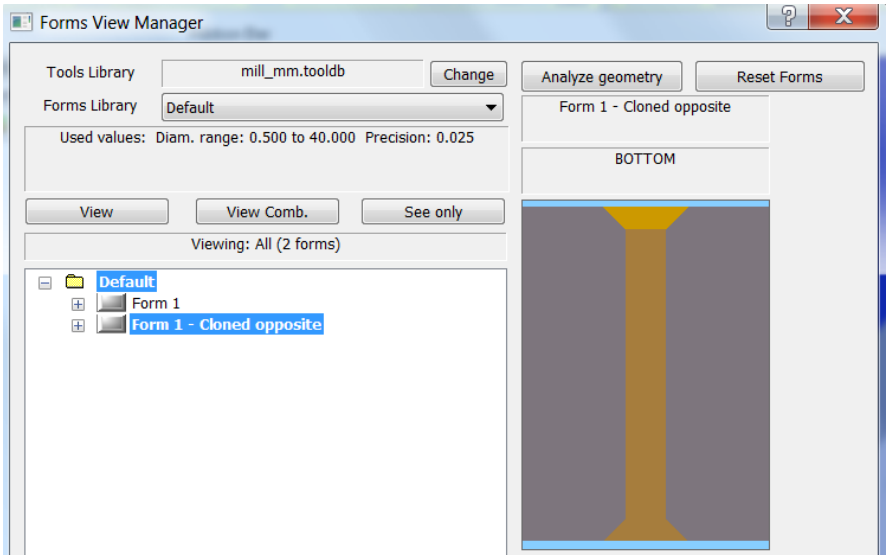


BOTTOM SIDE

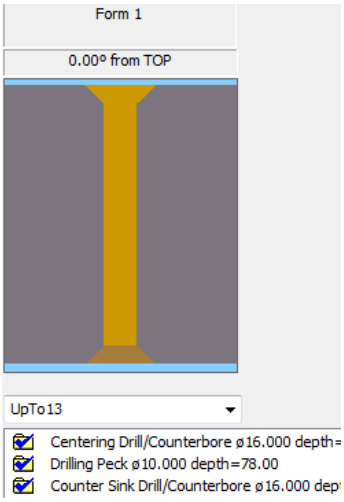


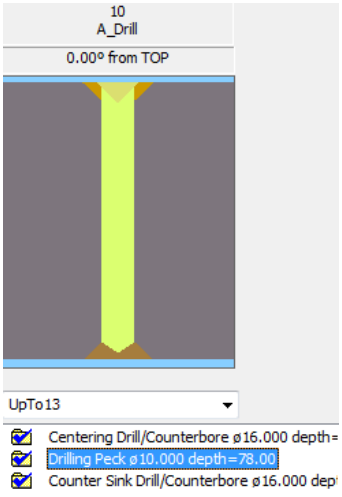
APPLYING PROCESSES AND STORING TO LIBRARY

Now that the forms are split, you can apply drilling processes to them. Highlight **Form 1** and click on "**Apply Automatic**"

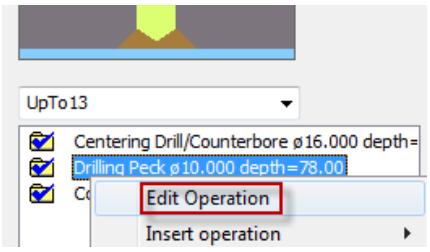


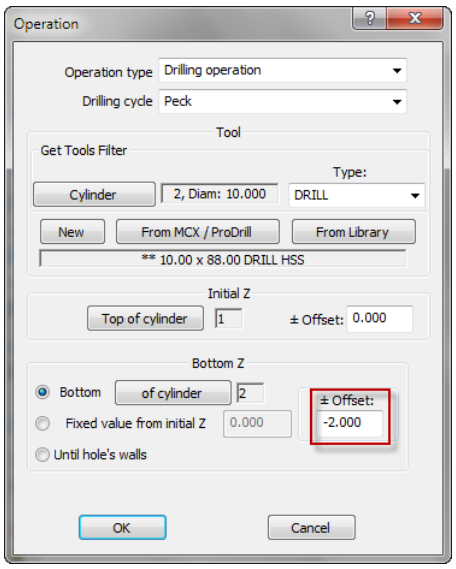
The following drilling processes are applied to Form 1. Upon simulating by selecting the drilling operation, you will notice that the drill depth is set to 78mm.



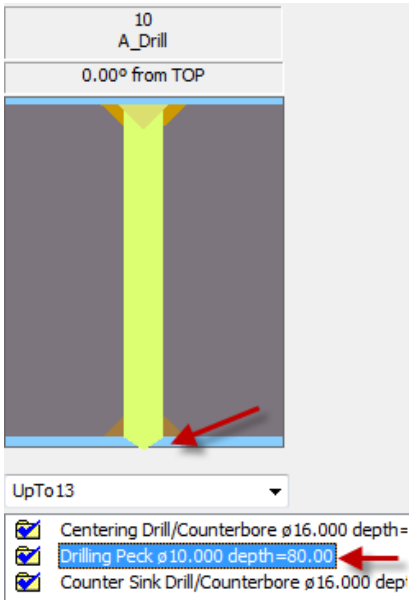


The depth is not sufficient to drill through the part if we are only going to apply the counter sink operation to the bottom side. We need to add an additional depth of 2 or 3mm to the process. Right click on the drilling operation and choose **Edit Operation**. For the Bottom Z depth, add a value of **-2mm** to the **offset** value.

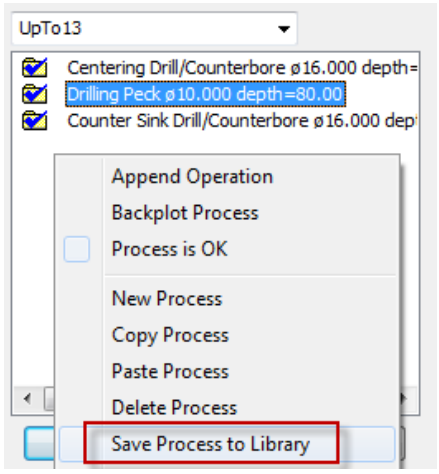




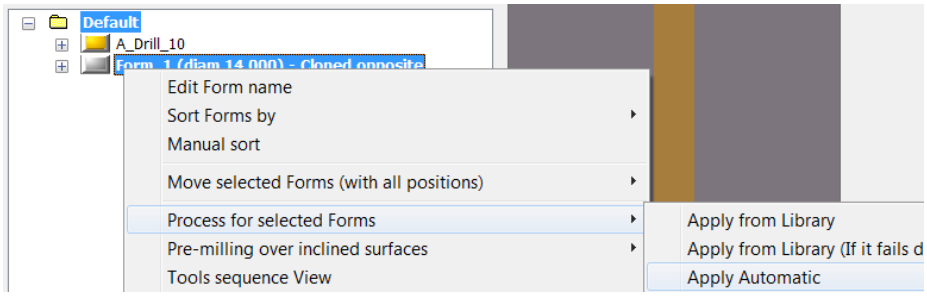
Click on **OK**. The resulting depth is now 80mm instead of 78mm. You will also note that the tool drills through the part.



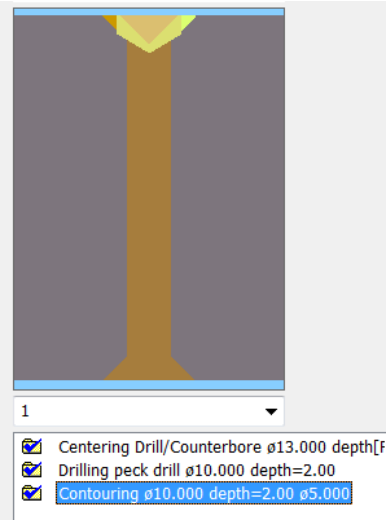
Now that you have customized the process, store the process to the Library by right clicking and choosing "**Store Process to Library**".



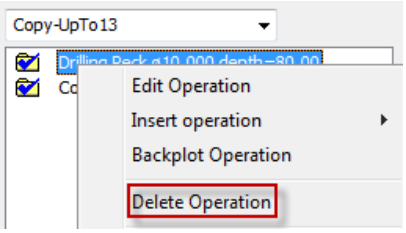
Now select **Form 2** from the tree and apply a process by selecting "**Apply Automatic**".



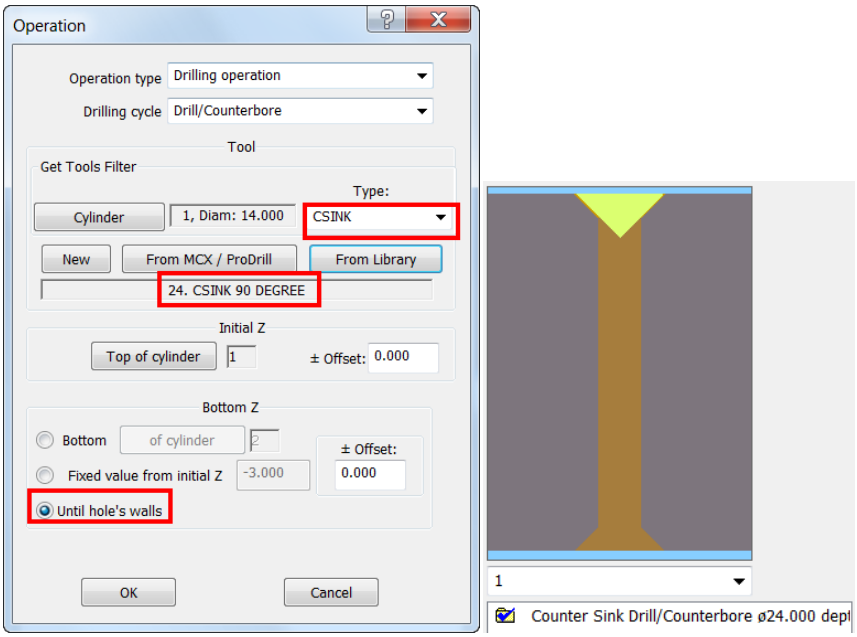
A new process is now added to the form. Note that in this case, all 3 drilling operations are added. To machine the chamfer, we only need a counter sink operation.



Delete the last 2 processes for peck drilling and contouring.



The result of modifying the process is only a counter sink operation to machine the chamfer on the opposite plane. Modify the operation as shown to machine the chamfer. Change the tool type, select a tool from library and change the depth setting to "Until hole's walls".

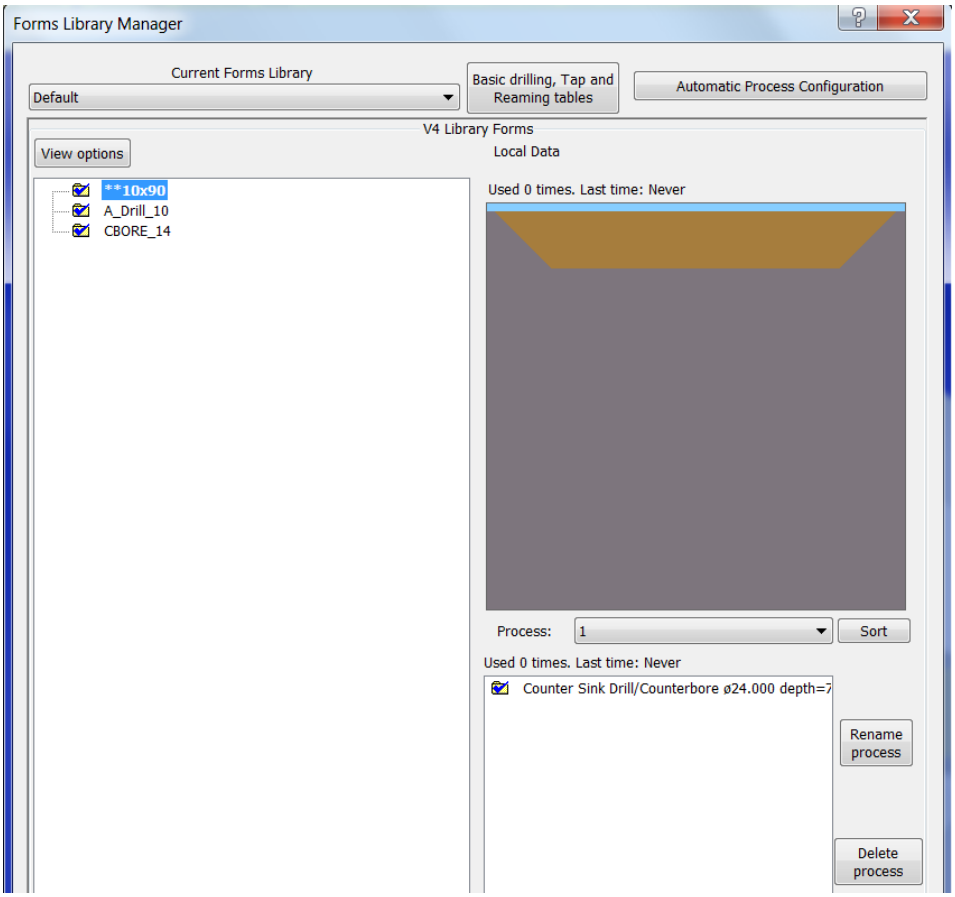


Now, store this new form to the library.

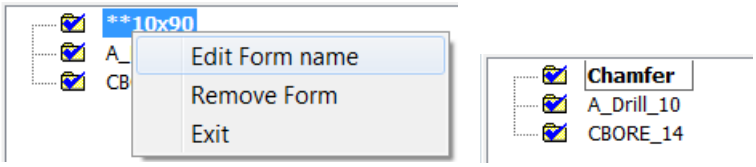
Close the Forms View Manager dialog.

MODIFYING PROCESS NAMES AND FORMS NAME IN LIBRARY

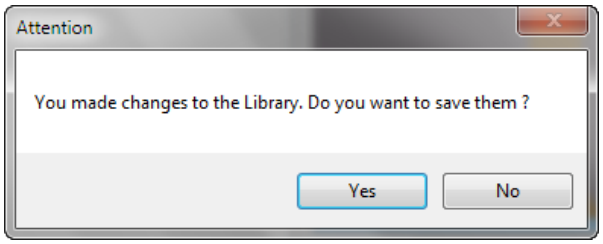
Open the **Forms Library Manager** dialog. Click on the **10X90 form.



Let us **rename** the name of the form **10x90 to Chamfer.** Right click on the **10x90 and select **Edit Form name**. The name is highlighted for editing. Enter the new name.

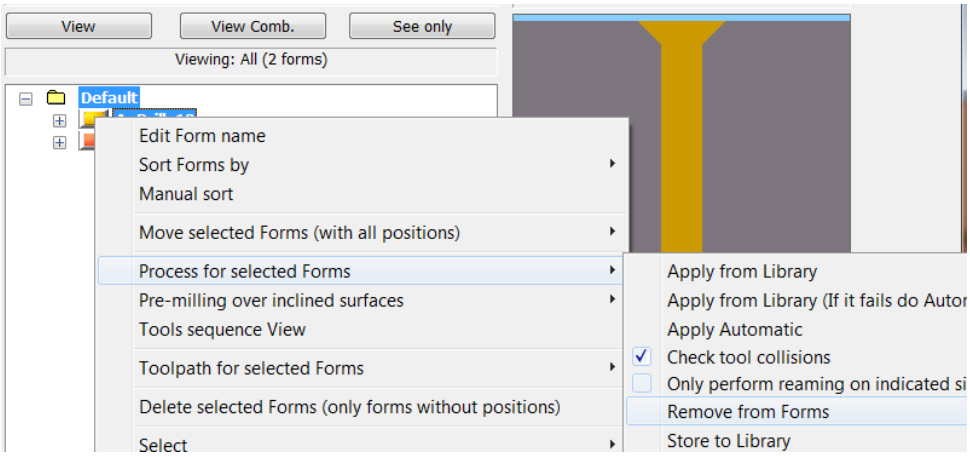


Now that you have renamed the processes and the form name, close the dialog by clicking on **OK**. You will be prompted to save the changes made to the library. Accept the changes by click on Yes.

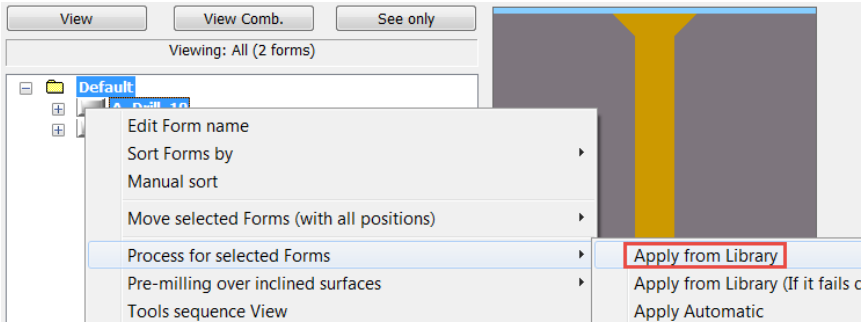


REAPPLYING PROCESSES FROM LIBRARY

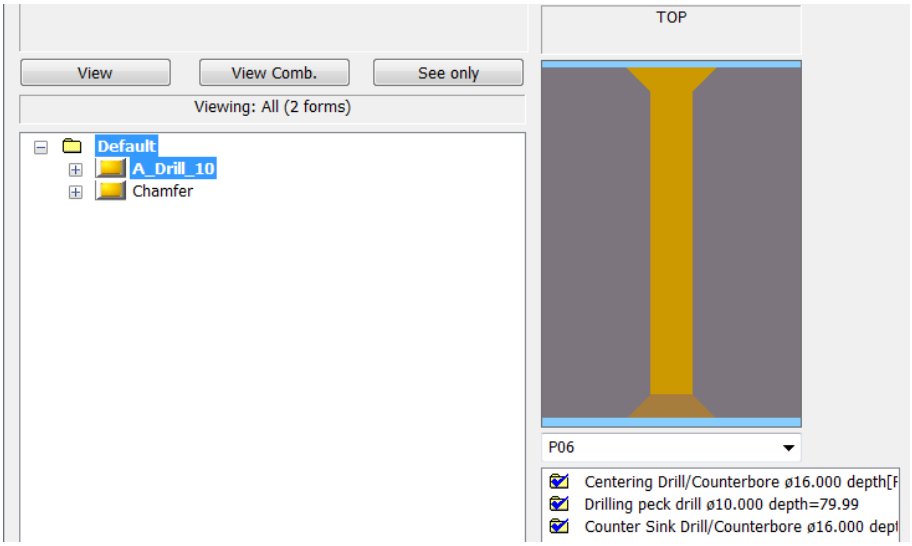
Now that we have edited the names, launch the **Forms View Manager** and remove the processes from the 2 forms.



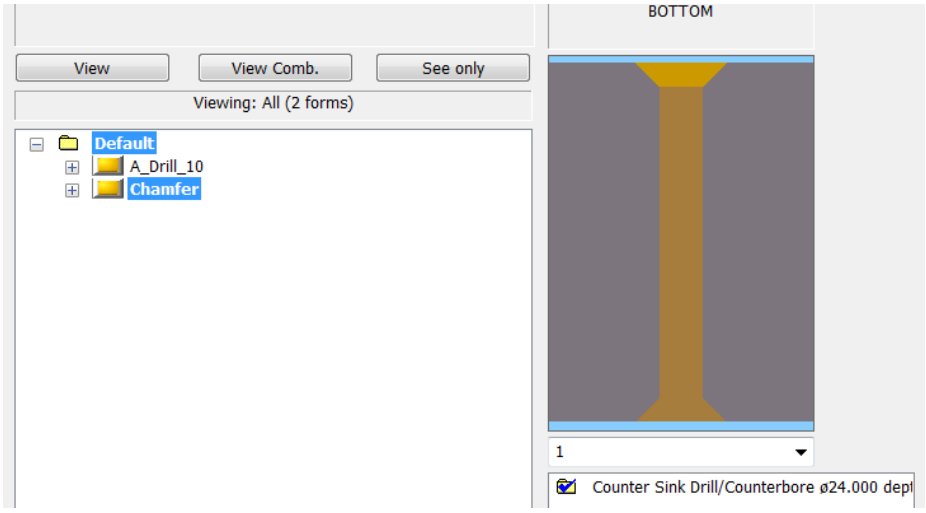
Now we can apply the library processes to both the forms. Highlight both forms and click on "**Apply from Library**".



The process names are now added as in the library and the same processes are added to both the forms.



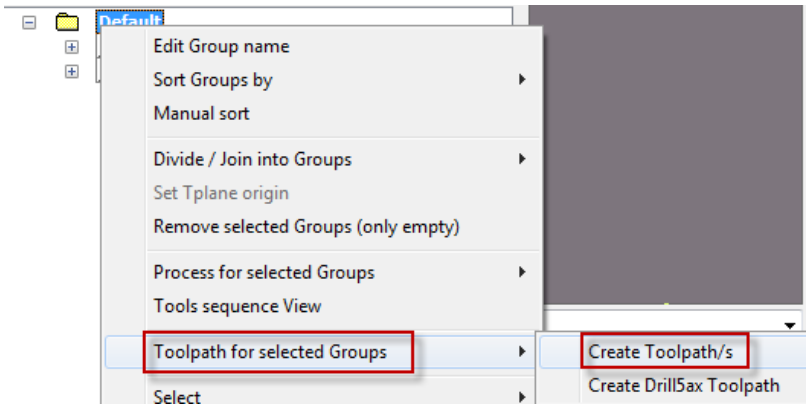
The second form that only has the chamfer has the Chamfer form applied with one operation.



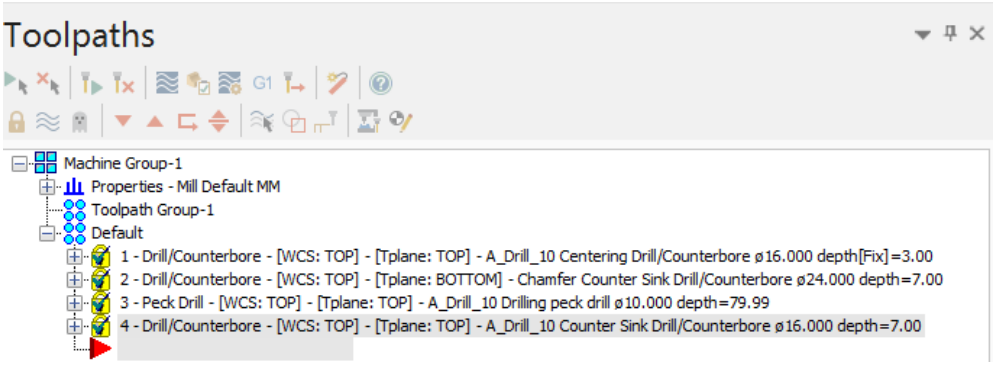
We have easily created new processes, copied them, renamed them and reapplied them for these forms.

CREATING TOOLPATHS AND GROUPING BY VIEWS

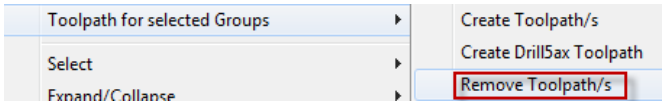
To machine these forms, we can right click and create toolpaths for them. Click on Default to select both forms, right click and create toolpaths.



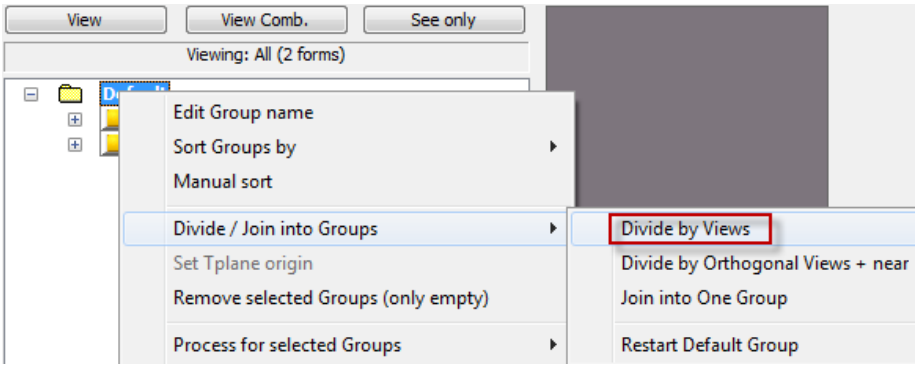
You will notice that the 2 drilling operations are done together. We do not want the counter sink operation (2nd operation) to be drilled for the bottom plane right after the centering for the top plane.



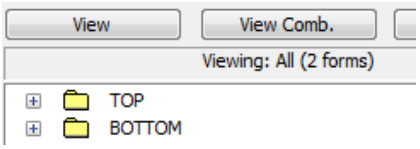
To split the forms by the views they are identified on, we can split the group by views. Return to the Forms View Manager, **highlight** the **Default** group, **Remove Toolpaths** for both forms.



Select the **Default** group, right click and select **Divide by views**.

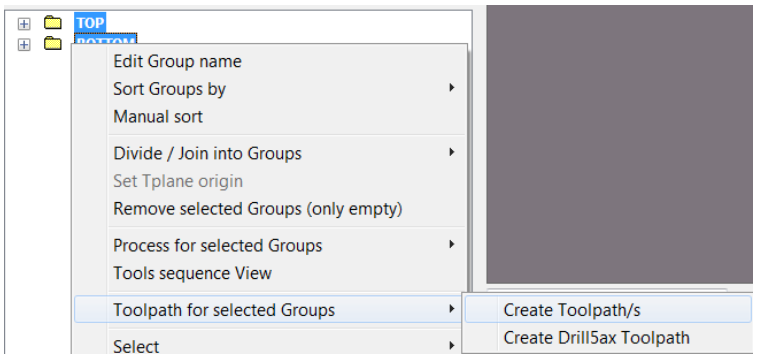


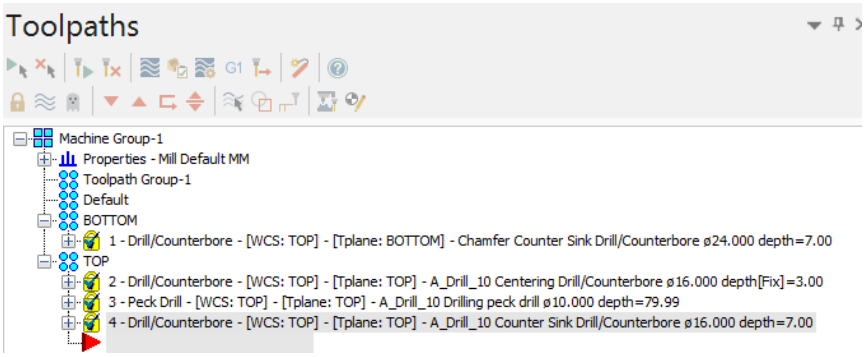
The result is 2 new groups according to the construction plane that the forms are identified on:



If you highlight the views, all the forms identified on the Top view are grouped together and all the forms on the Bottom view together (in this case the chamfers).

Create the toolpaths now for both groups. You will notice the toolpaths are created according to the planes.



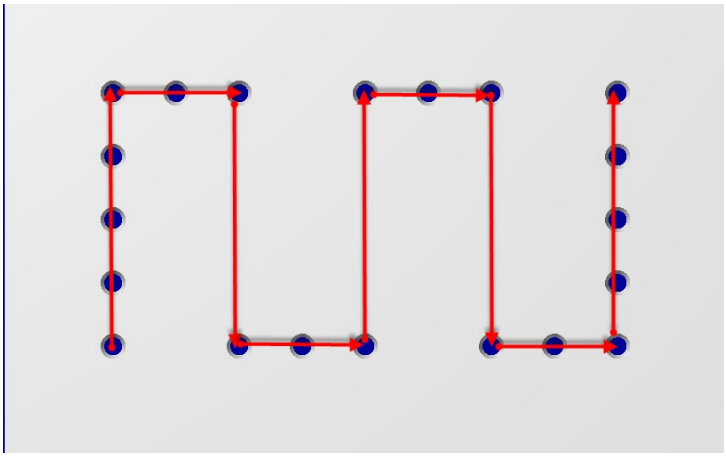


This concludes the exercise.

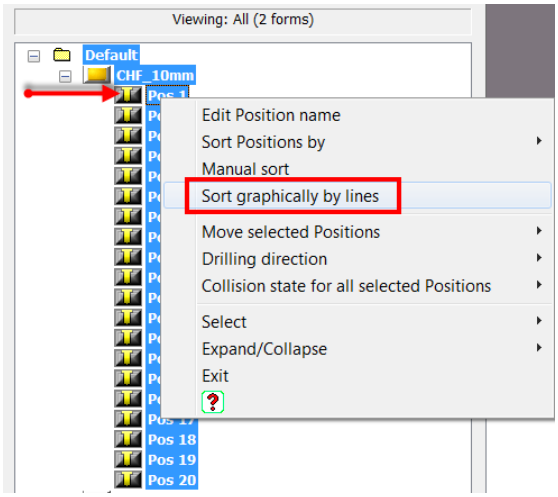
SORTING FORMS GRAPHICALLY BY SKETCHING LINES

Using the same part as the previous exercise, you can sort the drilling direction for the forms.

You will see that the holes are not sorted in the optimal manner for drilling. The following is the drilling direction.

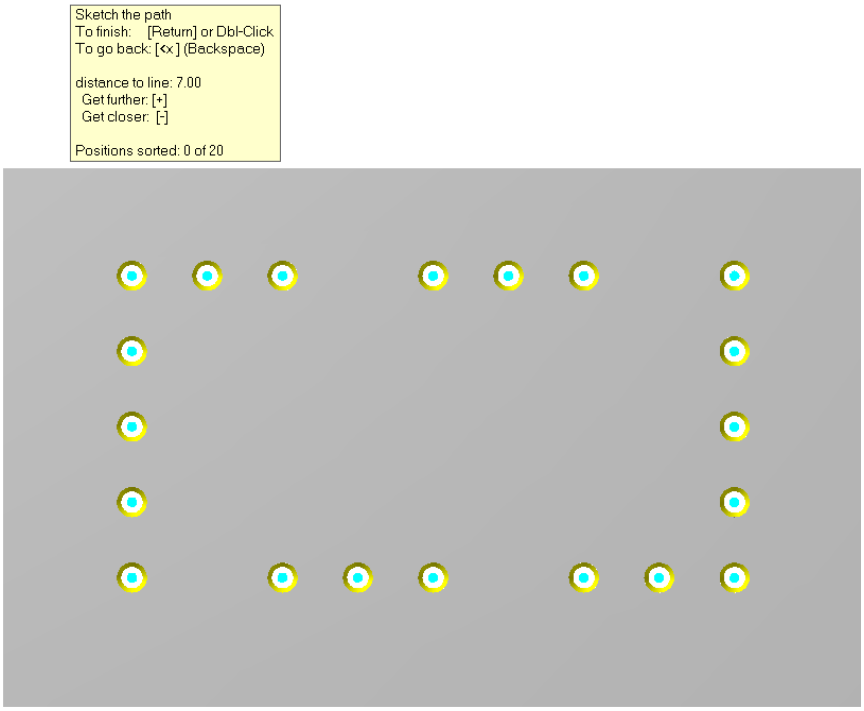


If you want to use a better sorting method for such parts, use the following option found in the right click option "Sort graphically by lines".

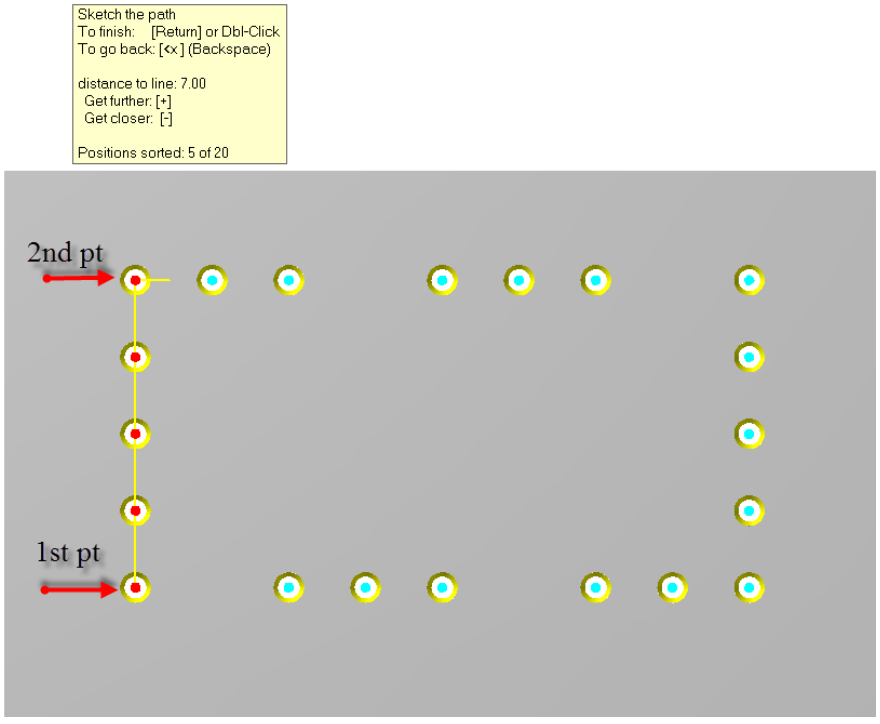


The following dialog is presented. You can now indicate the direction for starting and ending by clicking on the appropriate points.

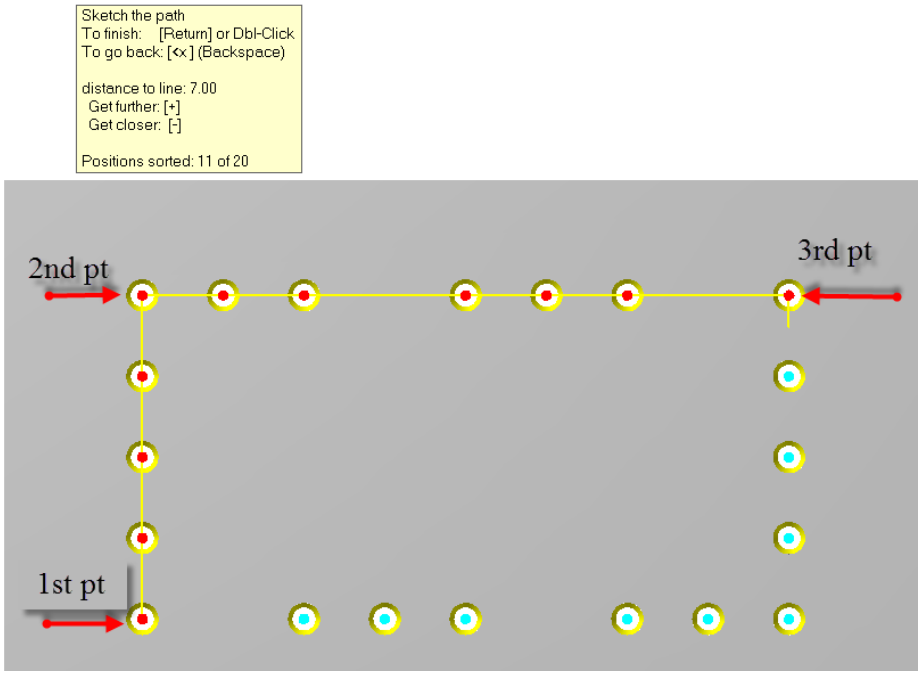
The messages indicate how to use the feature. There is a tolerance control for this feature as well. Currently, none of the 20 points are sorted (as indicated by the message).



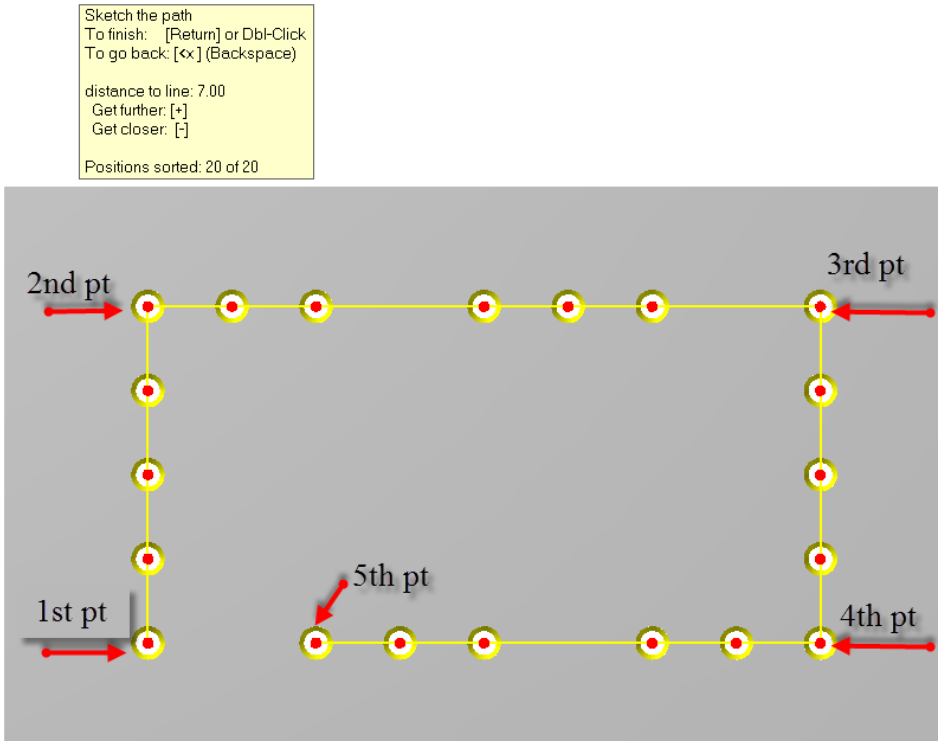
Click on the 1st point as indicated, then click on the 2nd point as indicated. A yellow line is sketched and all the points that are identified along that line are automatically highlighted with red points. The point count indicates that 5 of 20 points are sorted.



Now click on the 3rd point as indicated. The next set of points are selected. 11 of 20 points are selected.



You can sort the rest of the points by clicking on the following 2 points.



Click on **Return** or Double click to accept the sorting. The tree is now updated using the new sorting method.

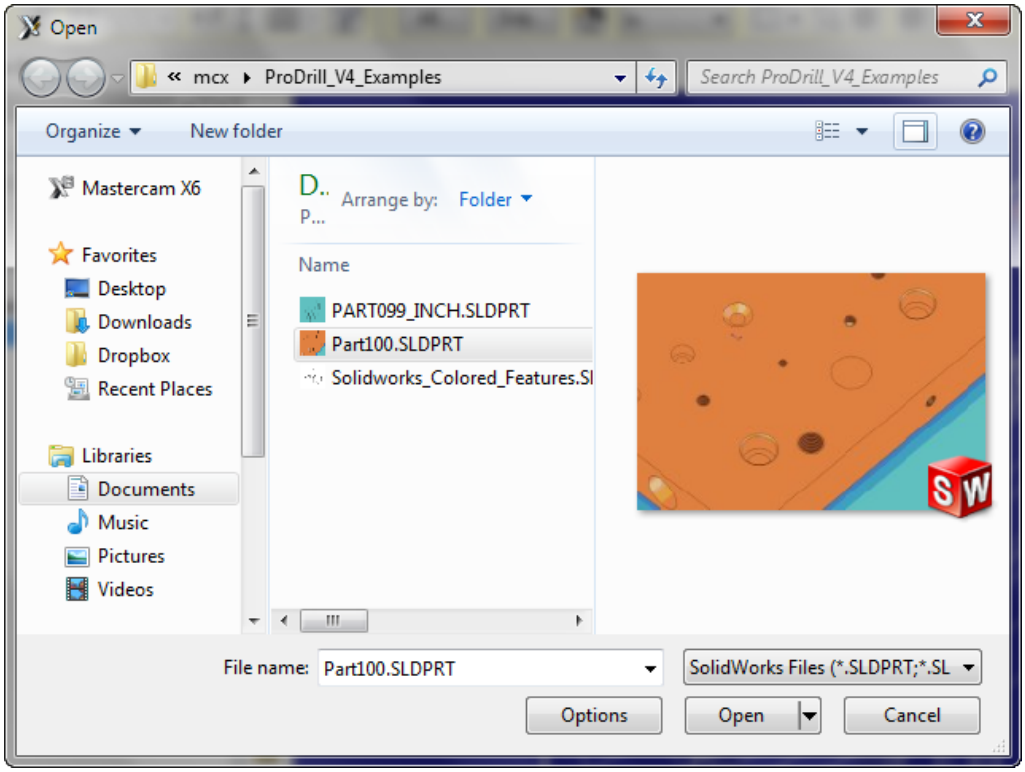
With 5 clicks, you have sorted the drilling direction!

IMPORTING REAMED AND TAPPED FEATURES FROM A SOLIDWORKS FILE

Many Solid modeling systems allow the creation of drill forms with tapped and reamed hole details. Importing these features into Mastercam via ProDrill is available for these systems.

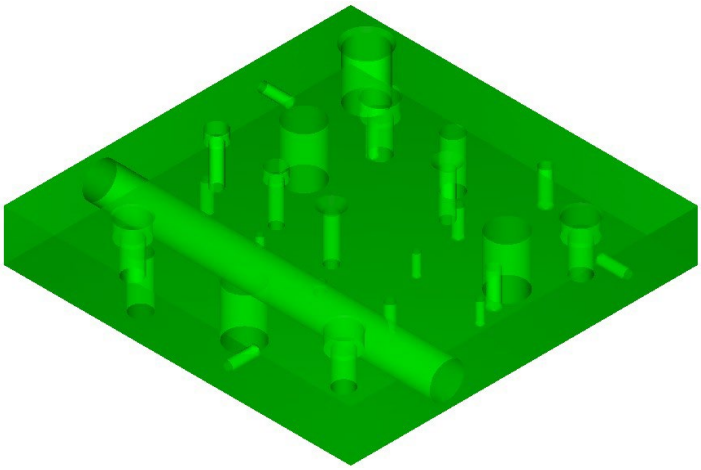
There is a SOLIDWORKS partfile provided in the ProDrill examples folder along with the corresponding feature data file generated from within SOLIDWORKS (using a Macro provided with the ProDrill installation).

Import the SOLIDWORKS file **PART100.SLDPRT** using the Mastercam File, Converters, ParaSolids import feature. Select the following file from the *ProDrill_V5_Examples* folder:



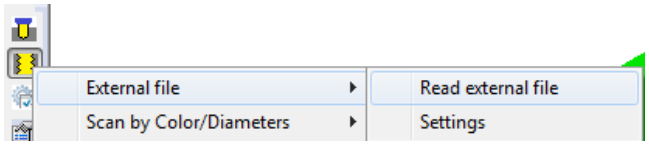
This is a METRIC file. Import it as a Solid or surfaces. Edge curves are not essential for ProDrill.

The following should be the result of the import:

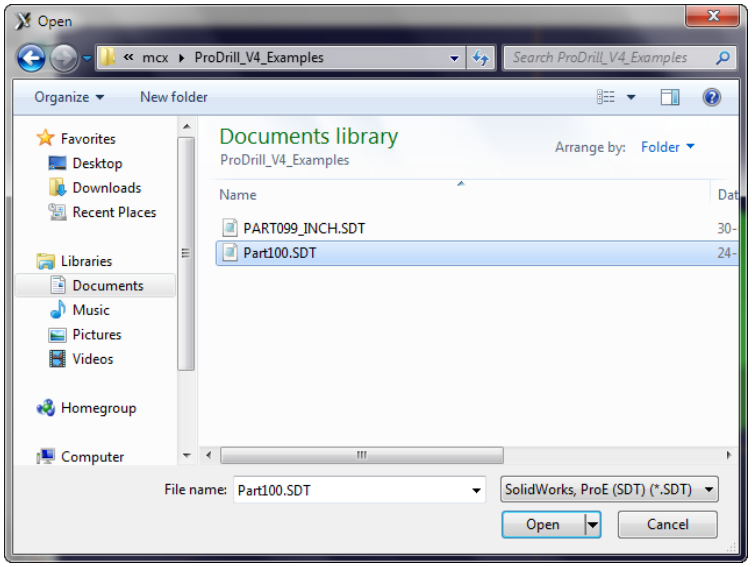


In order to mark the reamed and tapped forms, there is a drill feature data file that was generated using the Macro provided for SOLIDWORKS by ProDrill (review the Powerpoint presentation on using the Macro and generating the output file). This file has an extension .SDT that is normally stored with the same name and location as the SLDPRT file.

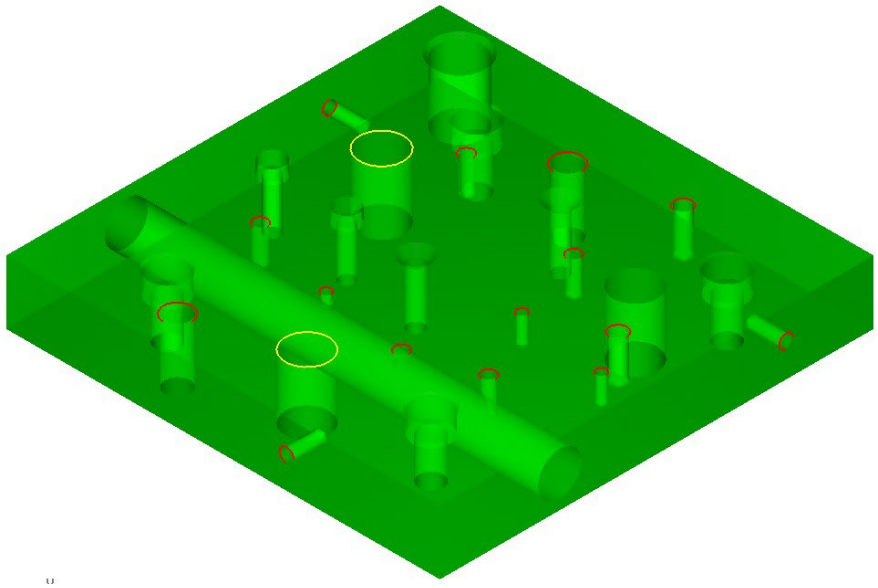
Click on **Set Features** and choose the following option:



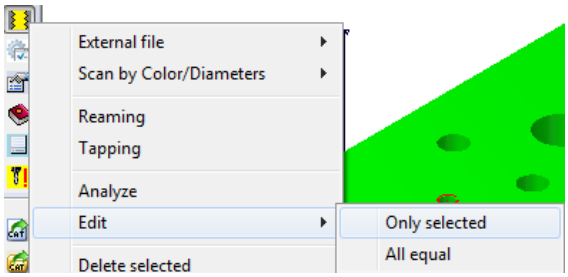
Select the PART100.SDT file from the same folder as the SLDPRT file.



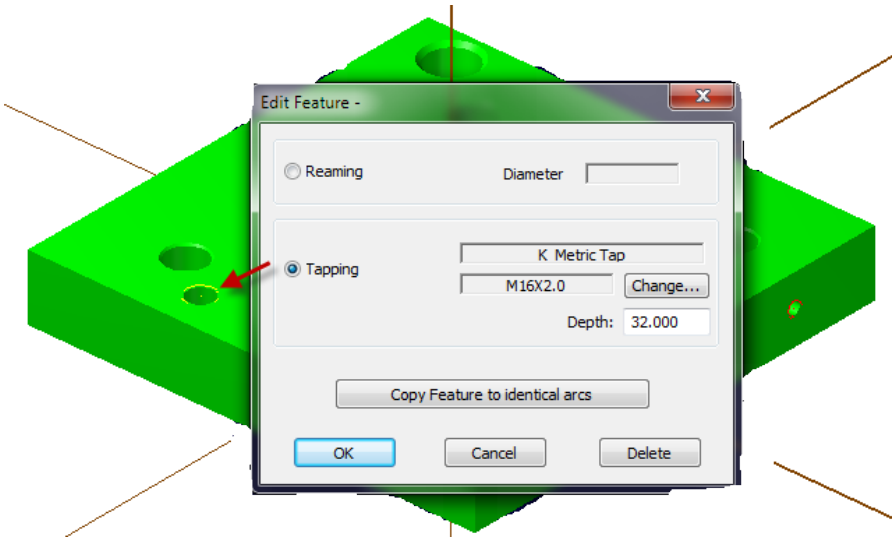
Select the file and click Open. The drilling data that contains the reamed and tapped hole information is imported into Mastercam. All the holes that need to be reamed or tapped are marked with the corresponding red or yellow arcs in the appropriate drill form locations!



Clicking on **Edit, Only Selected** will allow you to determine the tap sizes, etc.



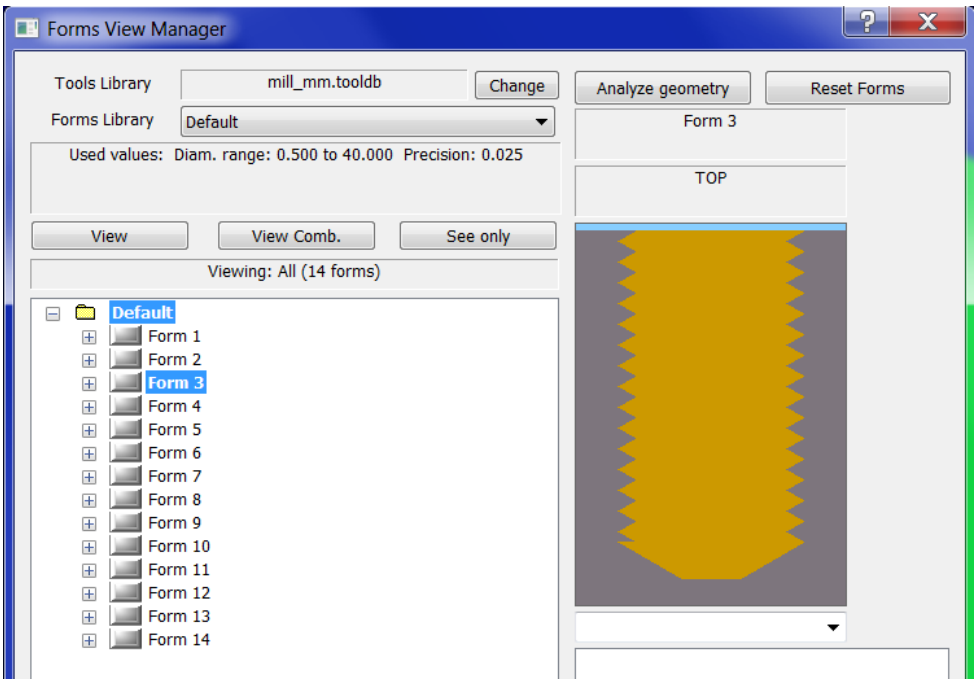
The following will be presented upon clicking on the following red arc:



Within a few clicks you have identified all the drill forms that need to be reamed or tapped from an imported file!

We can now process the file to see the result from within ProDrill.

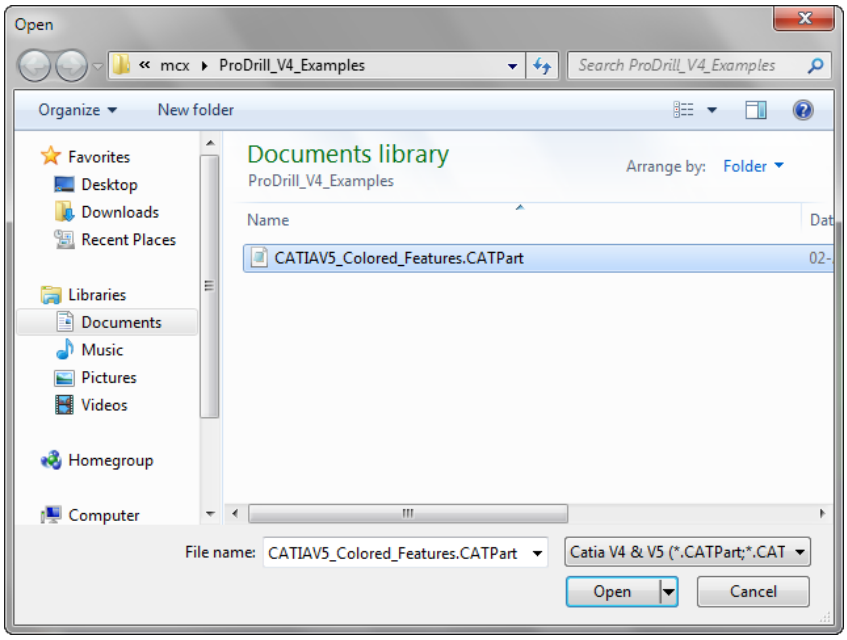
Click on **Forms View Manager** to process the part. All the tapped and reamed forms are readily identified in the part!



IMPORTING COLORED FEATURES FROM A CATIA FILE

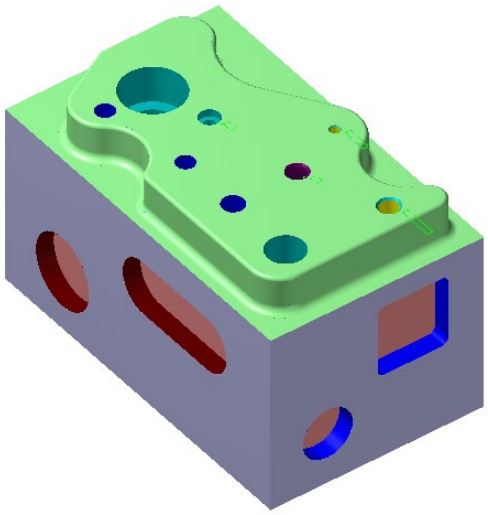
Importing reamed and tapped features that are assigned specific colors in external CAD systems such as CATIA, SOLIDWORKS or any other CAD solution can now be configured and easily processed with ProDrill.

IF you have the Moldplus CATIA translator, then set the option for the CATIA V5 file to import as surfaces or Solids and import the following file from the \ProDrill_V5\ ProDrill_Parts folder **CATIAV5_Colored_Features.CATPART**. This is a sample file designed with color coded specifications for one of the automobile manufacturers (BMW in this case).



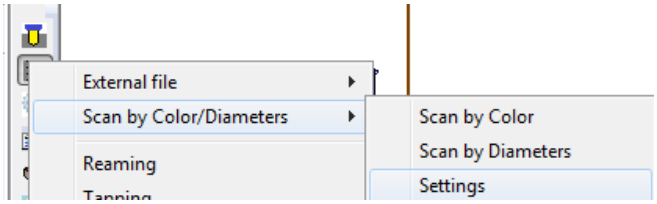
The following file is imported as surface or Solid data.

If you do not have the Moldplus CATIA translator, the MCAM file is available for you in the \ProDrill_V5_Parts\ folder called **CATIAV5_Colored_Features.MCAM** file. You can load this file if you cannot import the CATPART file.



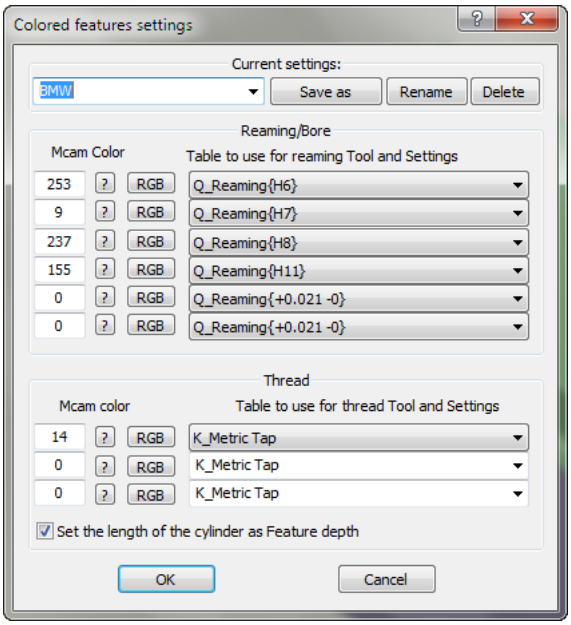
There are different cylinders marked with different colors for tapping and reaming. We have pre-defined color settings tables to determine the tapped and reamed sizes.

To set these features using ProDrill, click on the following:

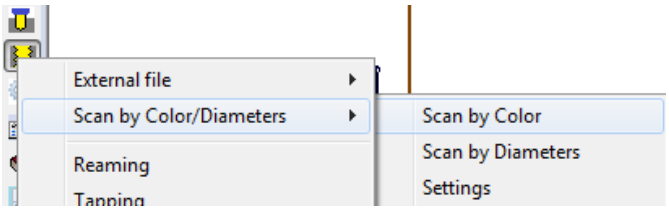


The colors are already assigned and mapped according to the RGB colors and corresponding Mastercam colors. You can customize your own color settings and create new configurations as needed.

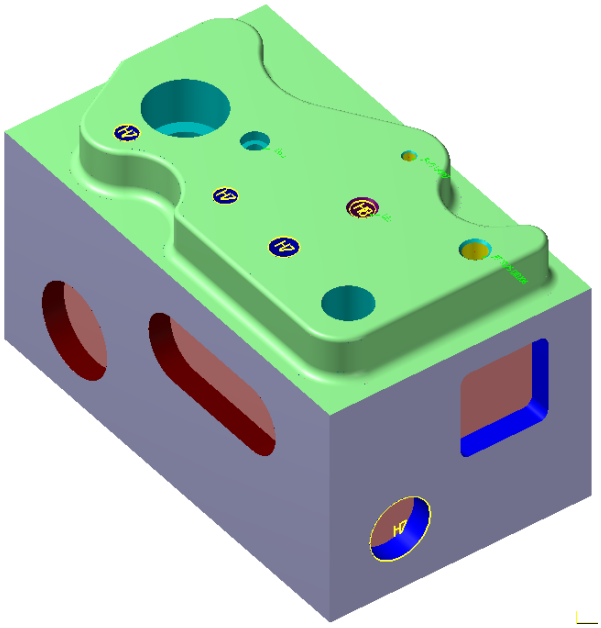
Choose the **BMW** settings option.



Click on **OK** and then click on **Scan by color**.



The part is processed according to the colors and corresponding reamed tolerance and tap tables and the forms automatically identified for reaming and tapping! In this case, H6, H7 and H8 reamed forms are identified along with M6, M12 and M14 tapped forms.



You can analyze the features if needed. You will notice that all the tapped forms were designed with the same color but ProDrill determines the tapped forms by scanning all the set colors with the internal diameters of the assigned tap table.

When you execute this part, the tapped and reamed forms will be processed!

There is also a SOLIDWORKS part - **Solidworks_Colored_Features.SLDPRT** for testing with colored reamed and tapped forms also in the same manner. For this part, you can choose the SOLIDWORKS Color Settings configuration to process the part with.

PRODRILL SUPPORT

ProDrill support is normally provided through your local Mastercam reseller.

When sending files for support, please include the following files:

1. Mastercam file of the geometry in question
2. ProDrill form library used to process the file (entire folder of the Forms library)
3. Mastercam tool library (.TOOLSDB) file used to process the file
4. Images of any issues that you would like to report

Thank you for purchasing ProDrill V5 for Mastercam!

Mastercam® is a product of CNC Software, LLC

SOLIDWORKS® is a product of Solidworks Corporation