

Purdue AFL

CATIA CAM Process Reference

Rev. B



Revision Notes

Revision - of this document refers to the CATIA v5R21 deployment of the AFL CATIA Environment. All information contained in this reference document does not pertain to earlier releases of the environment. The process catalog corresponding to this document is titled **AFL_Catalog_B20.catalog**.

Revision A of this document refers to the CATIA v5R21 deployment of the AFL CATIA Environment. All information contained in this reference document does not pertain to earlier releases of the environment. The process catalog corresponding to this document is titled **AFL_Catalog_B21.catalog**.

Revision B of this document refers to CATIA v5R21 deployment of AFL CATIA Environment. All information contained in this reference document does not pertain to earlier releases of the environment. The process catalog corresponding to this document is titled **AFL_Catalog_B21.catalog**.

Contents

- Revision Notes 2
- Revision History 4
- Process: Rough Cut 5
- AFL Prismatic Machining Processes: 7
 - Process: Pocketing After Rough Cut 8
 - Process: Pocketing After Rough Cut w/RWK 9
 - Process: Pocketing For Narrow Areas 10
 - Process: Narrow Areas w/RWK 10
 - Process: Drill Simple Hole 11
 - Process: Non-Standard C-Bore 12
 - Process: Counter Sink 15
 - Process: Standard Tap 16

Revision History

Rev	Revision Description	Date	Revised By
-	Initial Release	2/27/13	P. La Petina
A	Modifications made for move from CATIA v5R20 to v5R21 <ul style="list-style-type: none">- Added note after contents mentioning this document is specific to R21 and the B21 version of the AFL catalog.- Modified catalog naming convention- Renamed Rough Profiling process to Semi-Finish Profiling	5/6/14	P. La Petina
B	<ul style="list-style-type: none">- Added documentation on Standard Counter bore, Drill and Ream, Chamfering- Modified documentation on Tapping	7/10/14	S. Swaminathan

Process: Rough Cut

Chapter: 'AFL_Catalog/2.5/3D Roughing'

Overview:

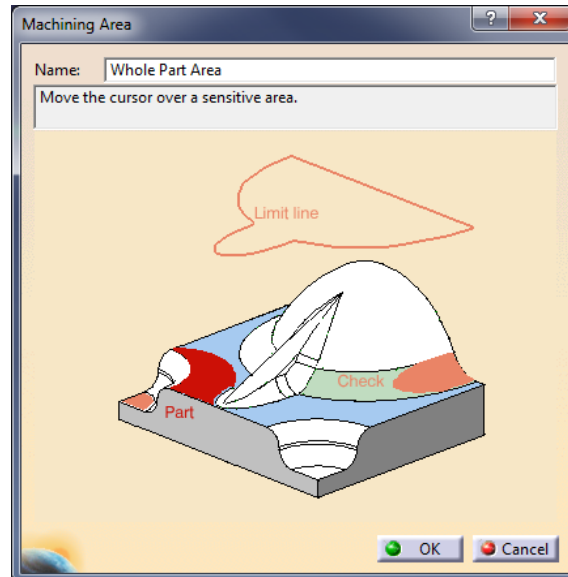
The *Rough Cut* process is a versatile process for removing the majority of the stock. It should be used first and be applied to the entire part when possible. The other processes are finishing processes and should only be applied after a rough cut.

Geometry Needed:

Rough Cut requires a 'Machining Area' to be created. This is a 3-D version of the 'Prismatic Machining Areas'. Choose the design part as the "Part" for the machining area. If the entire part is to be roughed (normally so) then no limit line is needed. If there is a chance that the vise or other fixturing will be hit by the tool, choose the faces of this fixturing as "Checks."

Detailed Description:

The *Rough Cut* process initializes with a 1/2 inch HSS end mill. Another tool can be used for large or small parts, consult the AFL. This process will rough out all the material within the limiting contour. The process cuts the part leaving .03 inches of stock on the design part. It will then cut all horizontal areas to within .01 inch depth. All settings are preset and do not require adjustments. This process should be applied first and normally only once. After the rough cut, the other processes will be applied in order to remove the remaining material and finish the surfaces.



Important settings:

The rough cut will try to rough holes larger than .6 inches. If this is not desired (if you are drilling the large holes) then change the parameter "Ignore Holes in Stock" to a diameter larger than your hole diameter (Roughing.1 - Geometry Selection Tabpage – Ignore Holes in Stock – Diameter).

If limiting contours are used, there are several settings that control what the contour actually means. These are found in the "Limit Definition" box on the geometry tabpage. Feel free to change these settings as needed.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.

2) All approaches should be made from the outside or be ramping approaches.

Common Problems:

A common problem is that the automatically selected tool axis will be wrong and not perpendicular to the operation's safety plane. To fix this problem, go to the strategy tabpage and select the green arrow in the upper right hand corner of the graphic (see below). Then a dialog box will open where you will set $i=0$, $j=0$, and $k=1$.

Other common problems arise when using limiting contours. Please consult the AFL if you have problems with your limiting contours.

AFL Prismatic Machining Processes:

The AFL provides three different prismatic machining processes. These are Pocketing After Rough Cut, Pocketing After Rough Cut w/RWK, and Pocketing After Rough Cut for Narrow Areas. Each of these processes is used to complete a pocket (a prismatic area) after a rough cut. The difference is in how much of the pocket cannot be cut with a ½ inch tool. Below is the details of how this decision is made

Use *Pocketing After Rough Cut* if:

- a) All inside corners of your pocket are allowed to have a .3 inch radius
- b) All areas of the pocket are wider than .5 inches with about .072 inches to spare. (Minimum channel width > .572)

Note: if you are unsure whether or not this is true, you can simply check the result of Rough Cut operation. If was able to rough the entire pocket then use the Pocketing After Rough Cut cycle. If not choose one of the other prismatic processes.

Use *Pocketing After Rough Cut w/RWK* (with rework) if:

- a) Some or all of your corners of your pocket have less than a .3 inch radius. (Note the process will select a tool based on the smallest radius corner in the prismatic machining area. If this is smaller than can be done with a 1/8 inch endmill, the tool will not be automatically selected. In other words, be careful of models with square corners.)
- b) Greater than 20% of pocket is wider than .5 inches with about .072 inches to spare.

Note: if you are unsure whether or not this is true, you can simply check the result of Rough Cut operation. If was able to rough any significant portion of the pocket then use the Pocketing After Rough Cut w/RWK cycle. If not choose one of the other prismatic processes.

Use *Pocketing for Narrow Areas* if:

- a) Some or all of your corners of your pocket have less than a .3 inch radius. (Note the process will select a tool based on the smallest radius corner in the prismatic machining area. If this is smaller than can be done with a 1/8 inch endmill, the tool will not be automatically selected. In other words, be careful of models with square corners.)
- b) Nearly the entire pocket is too small to mill with a ½ inch end mill. Great example are slots of width < .572 inches.

Note: if you are unsure whether or not this is true, you can simply check the result of Rough Cut operation. If it was not able to rough any significant portion of the pocket then use the Pocketing for Narrow Areas cycle.

Following this section is a complete description of the *Pocketing After Rough Cut* cycle. A shorter description of the other two prismatic cycles is given after. These two processes are just expanded versions of the first so this information is repeated.

Process: Pocketing After Rough Cut

Chapter: 'AFL_Catalog/Prismatic Processes'

Overview:

Pocketing After Rough Cut is designed to complete the machining of a pocket (a 2.5 D prismatic area). When *Pocketing After Rough Cut* is applied, three process are created. These three processes are designed to work together to provide high quality surface finishes.

Geometry Needed:

Pocketing After Rough Cut requires a prismatic machining area. Either contour types or pocket types can be used, but it is far more frequent to use the pocket type option.

Detailed Description:

Pocketing After Rough Cut initializes with a ½ inch HSS end mill. Another tool can be used for large or small parts, consult the AFL. However, other process exist for areas with small corners or small channels that are more versatile than using the *Pocketing After Rough Cut* a different tools. Three processes are applied to each prismatic machining area when *Pocketing After Rough Cut is intialized*. They are as follows are also available in the catalog as individual:

Rough Profiling of Pocket: A profiling pass used to semi finish the walls of the pocket. Leaves .009 inches of material on the walls. Linear to circular lead in/out.

Bottom Finishing of Pocket: A “Pocketing pass” used to finish the bottom of the pocket. Consists of one pass in z. Ramps into part and retracts axially.

Finishing Profile of Pocket: This finishes the side walls and completes the milling of the pocket. This process use “Cutter Compensation” to account for the difference in end mill nominal programmed diameter and the actual measured diameter of the tool. But it uses zero diameter cutter comp – this is important to know when running the program on a mill!! Please remind the employee assisting you that the program uses zero diameter cutter comp not full diameter cutter comp. If any questions arise about this, consult with a grad TA or a supervisor. Approaches and retracts are the same as in rough profiling.

Important settings:

One setting that may need to be changed is the overhang percentage in the bottom finishing process (Strategy tab - Radial – Percent Overhang). This may be increased to as much as 100% if the pass leaves areas unfinished near soft boundaries.

For high quality surface finishes, the axial depth of cut for the Finish Profile can be adjusted. But only do so with the permission of the AFL.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.
- 2) Check all approaches and retracts. Specifically check that the profiling approaches and retracts do not interfere with opposite wall

Common Problems:

Most problems that occur involve improper geometry selection. Often island contours are accidentally selected and cause problems or unexpected tool paths. Watch the video tutorials on this subject and if further questions exist, consult the AFL.

Process: Pocketing After Rough Cut w/RWK

Chapter: 'AFL_Catalog/Prismatic Processes'

Overview:

Pocketing After Rough Cut w/RWK is designed to complete the machining of a pocket (a 2.5 D prismatic area) that needs a smaller tool to reach all areas. When *Pocketing After Rough Cut w/RWK* is applied, up to six processes are created. These processes are designed to work together to provide high quality surface finishes.

Geometry Needed:

See *Pocketing After Rough Cut*.

Detailed Description:

Pocketing After Rough Cut w/RWK automatically selects the tools it will need. If the pocket has square corners the tool selection will fail and the user will have to manually select the tool. The cycle involves six individual processes if the channel has both narrow areas and tight corners.

Important settings:

See *Pocketing After Rough Cut*.

The Desired Result:

See *Pocketing After Rough Cut*.

Common Problems:

See *Pocketing After Rough Cut*.

Process: Pocketing For Narrow Areas

Chapter: 'AFL_Catalog/Prismatic Processes'

Overview:

Pocketing For Narrow Areas designed to complete the machining of a pocket (a 2.5 D prismatic area) that needs a smaller tool to reach all areas. When *Pocketing For Narrow Areas* is applied, three processes are created. These processes are designed to work together to provide high quality surface finishes.

Geometry Needed:

See *Pocketing After Rough Cut*.

Detailed Description:

Pocketing For Narrow Areas automatically selects the tools it will need. If the pocket has square corners the tool selection will fail and the user will have to manually select the tool. The cycle involves three individual processes. The first should remove the majority of the material in the pocket, the second will finish the bottom and the third will finish the sides. This process should calculate the size of the lead in/outs to prevent collisions.

Important settings:

See *Pocketing After Rough Cut*.

The Desired Result:

See *Pocketing After Rough Cut*.

Common Problems:

See *Pocketing After Rough Cut*.

Process: Narrow Areas w/RWK

Chapter: 'AFL_Catalog/Prismatic Processes'

Overview:

Narrow Areas w/RWK is identical to *Pocketing For Narrow Areas* except that it includes corner rework. Same idea above for selecting RWK vs. regular

Process: Drill Simple Hole

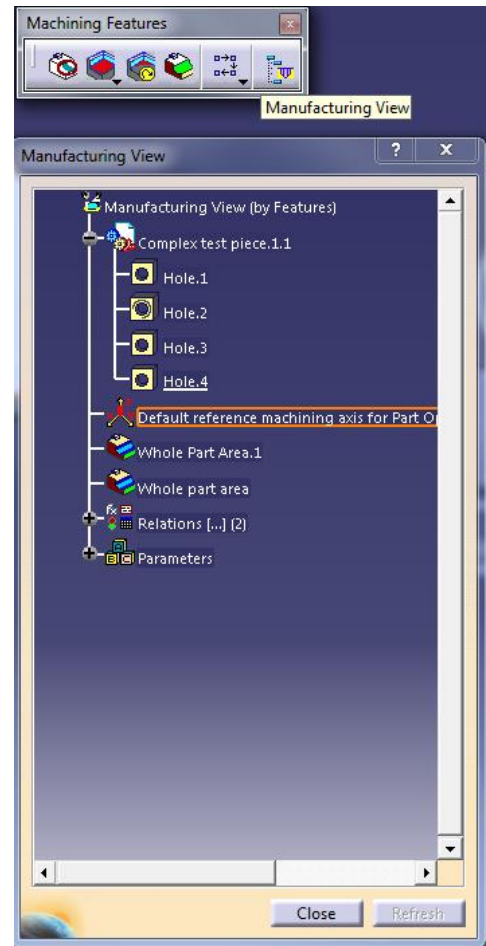
Chapter: 'AFL_Catalog/Axial Processes'

Overview:

Drill Simple Hole is the first process to applied to holes. It can then be followed by counter sinking, counter boring, tapping, etc. This process includes spot drilling.

Geometry Needed:

If the part was made in CATIA, then no extra steps are needed to create geometry. Simply open the 'Manufacturing View' (icon is in the same tool bar as the machining areas). Right click 'Manufacturing View' in the pop up window and choose 'sort by features.' The 'Manufacturing View' window should look something like the one to the right. Expand the tree of the design part and all holes in this part become apparent (ones that where modeled as holes in CATIA). If you have circular features that are the result of pads or pockets, see the Appendix for instructions about automatic feature recognition. You can simply choose these holes in the Manufacturing View when selecting geometry in the 'Process Application' dialog box. If there are patterns of holes, CATIA represents all of them by just one hole in the Manufacturing View. Creating 'Machining Patterns' is not advised as you instantly loss fidelity.



Detailed Description:

Drill Simple Hole initializes two processes. These are *Drill Deep Hole* and *Spot Drilling*. Spot drilling is very important to ensure that the drill does not jump when starting and ensures accurate hole placement. The *Drill Deep Hole* cycle uses a peck drilling canned cycle and will ensure that chips get removed. The process will automatically select a drill that is within .002 of the design size. Therefore do not use the automatically select drill if reaming is to occur. Also always check to make sure the intended drill was selected.

Important settings:

There are no setting that need to be changed by the user.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.

- 2) Check all approaches and retracts.
- 3) Check that the correct drill was selected.
- 4) Check that the video result's holes are the right depth and that through holes are completely thorough.
- 5) Check that the drill approaches the part from the correct side and that the drill is in the correct orientation.

Common Problems:

Sometimes CATIA will select the wrong direction to drill from. This can be seen from either tool path replay or the video simulation. The solution is to go to the geometry tab page and click the arrow above the hole. This will flip the orientation of the drill bit.

Most problems that occur involve improper hole depths or starting planes. If problems come up with this, consult the AFL.

Process: Drill and Ream

Chapter: 'AFL_Catalog/Axial Processes'

Overview:

Drill and Ream may be directly to holes. It can then be followed by counter sinking, counter boring, tapping, etc. This process includes spot drilling, peck drilling followed by reaming.

Geometry Needed:

See drill simple hole.

Detailed Description:

Drill and Ream initializes three processes. These are *Spot Drilling*, *Drill Deep Hole & Reaming*. Spot drilling is very important to ensure that the drill does not jump when starting and ensures accurate hole placement. The *Drill Deep Hole* cycle uses a peck drilling canned cycle and will ensure that chips get removed. The process will automatically select a drill that is .015" smaller than of the design size. This is followed by the reaming process. Also always check to make sure the intended drill was selected. The reaming process will plunge to full depth where it dwells for five seconds at full depth before retracting.

Important settings:

There are no setting that need to be changed by the user.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.

- 2) Check all approaches and retracts.
- 3) Check that the correct drill was selected.
- 4) Check that the video result's holes are the right depth and that through holes are completely thorough.
- 5) Check that the drill approaches the part from the correct side and that the drill is in the correct orientation.

Common Problems:

On occasions, CATIA will select the wrong direction to drill from. This can be seen from either tool path replay or the video simulation. The solution is to go to the geometry tab page and click the arrow above the hole. This will flip the orientation of the drill bit.

Most problems that occur involve improper hole depths or starting planes. If you encounter this problem please consult an AFL TA.

Process: Standard Counter bore

Chapter: 'AFL_Catalog/Axial Processes'

Overview:

Standard Counterbore is used to counter bore all holes.

Geometry Needed:

See *Non-Standard C-bore*.

Detailed Description:

This process uses the two flute counter bores. Standard counterbores are available in the AFL however any counterbore provided by the client may be modeled in CATIA and used with this process. If applied to all holes, this process will only initialize on holes with counterbores. The tool dwells for five seconds at the counter bore depth. Peck is not used in this process.

Important settings:

There are no settings that need to be changed by the user.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.
- 2) Check all approaches and retracts.
- 3) Check that the counter sink chosen is the right angle and diameter
- 4) Check that the result is the right depth.

Process: Non-Standard C-Bore

Chapter: 'AFL_Catalog/Axial Processes'

Overview:

Non-Standard C-bore is a milling process used to make counter bores that are different sizes than the standard counter boring tools in the AFL. This process will use an endmill to mill a circular pocket.

Geometry Needed:

If the part was made in CATIA and the holes are defined as being counter bored in the hole definition, simply select the hole from manufacturing view. For more information see the 'Geometry Needed' section of the *Drill Simple Hole* process. Note you can simply select all the holes, and the counter bore process will only be applied to those holes defined as counter bored. If this feature is not useful, use the "no-checks" catalog. If the part was not designed in CATIA, consult the AFL for more directions.

Detailed Description:

Non-Standard C-bore uses a helical milling process to mill the counter bore. The process automatically selects the proper sized endmill and will helical mill the counter bore.

Important settings:

There are no settings that need to be changed by the user.

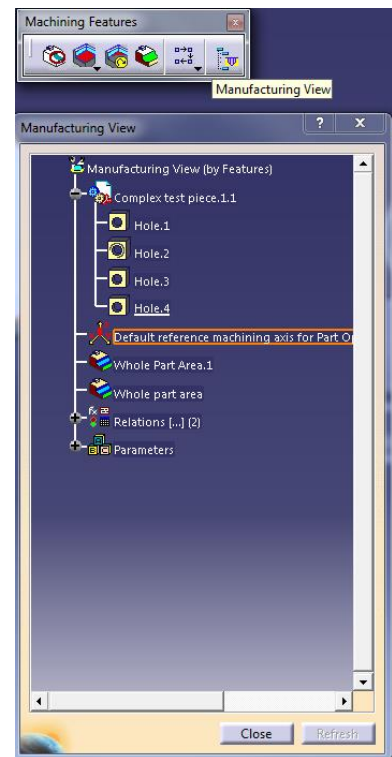
The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.
- 2) Check all approaches and retracts.
- 3) Check that the end mill selected is smaller than the hole.
- 4) Check that the result is the right depth and diameter.

Common Problems:

Most problems that occur involve improper counter bore depths or start planes. If problems come up with this, consult the AFL.



Process: Counter Sink

Chapter: 'AFL_Catalog/Axial Processes'

Overview:

Counter Sink is used to countersink all holes.

Geometry Needed:

See *Non-Standard C-bore*.

Detailed Description:

This process uses the single flute countersinks. Two angles are available in the lab but if tooling is provided by the client, any countersink can be used. If applied to all holes, this process will only initialize on holes with countersinks. No peck cycle used.

Important settings:

There are no settings that need to be changed by the user.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.
- 2) Check all approaches and retracts.
- 3) Check that the counter sink chosen is the right angle and diameter
- 4) Check that the result is the right depth.

Common Problems:

Most problems that occur involve improper counter sink depths or start planes. If problems come up with this, try choosing the “restore associatively” option in the geometry selection tab. If problems continue to persist consult with the AFL.

Process: Standard Tap

Chapter: 'AFL_Catalog/Axial Processes'

Overview:

This process is used to perform rigid tapping. **Much care must be given when using this process!!** The AFL approves only through holes to be rigid tapped. Consult the AFL about tapping blind holes.

Geometry Needed:

See *Non-Standard C-bore*.

Detailed Description:

Standard Tap will select the tap that matches the thread parameter of the hole. If the needed tap is not in the AFL standard milling library, simply model the tap in the document, taking great care to ensure all parameters are correct. Specifically, lead length and threads per inch (TPI). If the TPI is wrong **the tap will be broken off in the part!!!** Also note, broken taps are often impossible to remove. If questions arise, you are encouraged to consult with the employees in the AFL. This process will attempt to tap the entire thread depth, so be careful, most of our taps can only be used up to 1 inch deep. Check that the selected depth is possible with our tooling.

Important settings:

There are no settings that need to be changed by the user. But, if you model your own tap, make sure the tool parameters are correct. Be sure that the TPI is correct.

The Desired Result:

To verify the process, replay the tool path and check the following:

- 1) All horizontal rapid movements are above the part. NO horizontal rapid movements are allowed below the top of the part. This is AFL policy, and exceptions will not be made. This policy applies to all processes.
- 2) Check all approaches and retracts.
- 3) Check that the tap is the correct one.
- 4) Check that the machining feedrate is equal to $1/TPI$. Also have an employee help you check this in the final NC code.
- 5) Check the depth.

Common Problems:

TPI,TPI, TPI, make sure it is right! Also, make sure of #4 above. Problems with coding other than improper inputs are rare. Consult with the AFL to decide whether rigid tapping is possible and appropriate.