

CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™





CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™

What are Dynamic Machining Strategies™?

BobCAD-CAM, a well known CAD-CAM vendor has recently developed Dynamic Machining Strategies. DMS™ functionality allows users to apply any number of machining operations to a single CAD model feature,

as well as allow on-the-fly editing of these operations. This is where the user has the ability to load a single machining operation or multiple machining operations into one strategy "template". Like Pocketing or Profiling that has been assigned to CAD geometry/features. DMS is best used within a machining wizard. The machining wizard then stores the data and uses it for the creation of toolpath. DMS functionality is developed into

existing Machining Wizards of the cnc software to remove the guesswork of where the user is in the process, how to properly input the data for each machining strategy without missing critical details. This provides the maximum amount of efficiency in the programming workflow.

The bottom line is that Dynamic Machining Strategies exist to save the user time. Typically CAM software uses a Job/CAM Tree that is populated with machining strategies and operations.

Each time a machining strategy (2 Axis Milling, 3 Axis Milling etc.) is chosen, the new operation (Pocketing, Profiling, Facing etc.) is loaded into the tree. Each of these operations has tool data, speeds and feeds, lead-in and lead-out data, patterns, toolpath linking data and much more associated with it. In order for the CAM software to be fully "associative" the right CAD geometry has to be selected and assigned to an operation.

This means that some parts that have a lot of machining operations will have a lot of geometry associated with them and that the user has to stop and select geometry for every operation. DMS simplifies this process where the user simply picks the CAD geometry one time for the strategy being used and then the user loads all the different operations into the chosen strategy. The time savings is in not having to repetitively select geometry for every individual machining operation in the CAM Tree. Therefore, it's a faster, smoother process. In addition, typically the CAM Tree will look less cluttered which tends to help make programming less confusing in general.

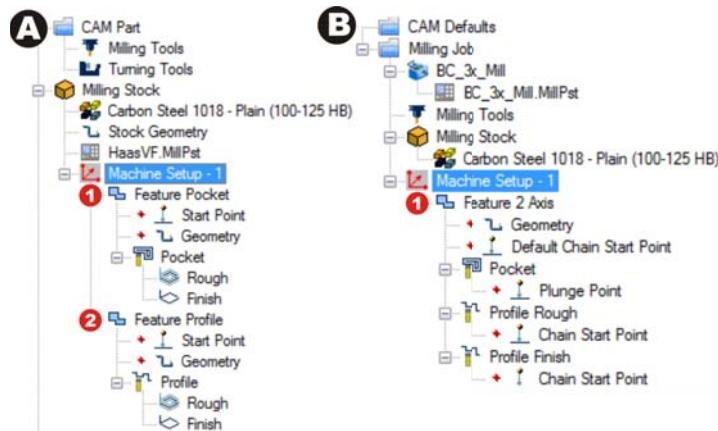
In order to see the full benefit of Dynamic Machining Strategies, there should be a basic understanding of how a CAM Tree manager flows. More specifically, how machining operations are loaded into the tree. Look at the two CAM Trees as shown (A and B). Both of them are for the same job.

The job is a simple milling type part that requires a:

- Pocket
- Profile Rough
- Profile Finish



CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™



CAM Tree "A" shows each machining feature loaded into the tree. Under each machining feature is a series of sub-that are associated to the machining feature.

Under Feature Pocket there is the ability to choose a start point as well as the associated geometry to the pocket feature. Then again Pocket is listed, which is where editing the strategies takes place. Finally there is Roughing and Finishing toolpath operations. That is a lot to have for one strategy. CAM Tree "A" also includes an unnecessary Lathe item (Turning Tools) in the tree when the job being programmed in the example is for Mill.

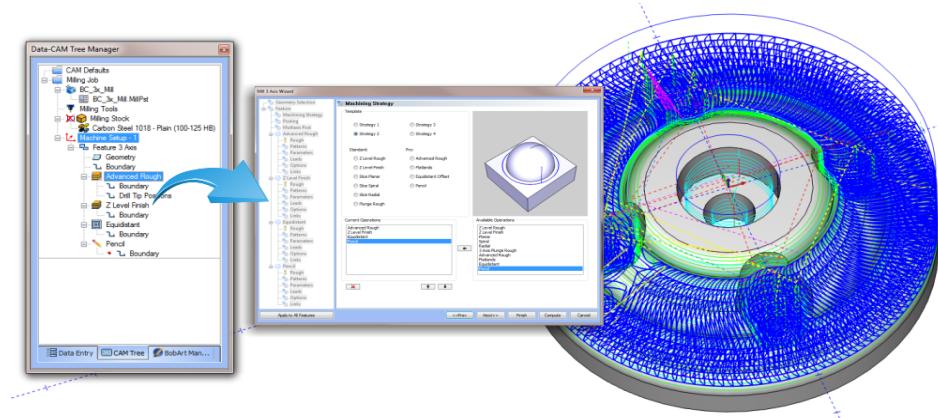
Now look at CAM Tree "B". This is a more consolidated and efficient CAM Tree. There is one strategy that contains the three machining operations within it. The Machining Wizard steps the user through the process of editing the 2 Axis strategies which include pocketing, profile roughing and finishing. That means the tools, the toolpath patterns, cutter compensation for the profile, step over and roughing step-downs etc are all set within the strategy wizard for each operation. CAM Tree "B" also offers a customizable Start Point for each machining operation. There isn't the need to keep selecting geometry in CAM Tree "B" because the user selected the CAD feature and then loaded all the machining operations associated to the geometry/feature. In this example CAM Tree "B" is more efficient.

When it comes to 3 Axis machining there can be scenarios where there are more than two machining operations required.

In this example there is:

- Advanced Roughing
- Semi-Finishing
- Finishing
- Pencil

Here is what this type of CAM Tree should look like in a 3 Axis machining scenario.



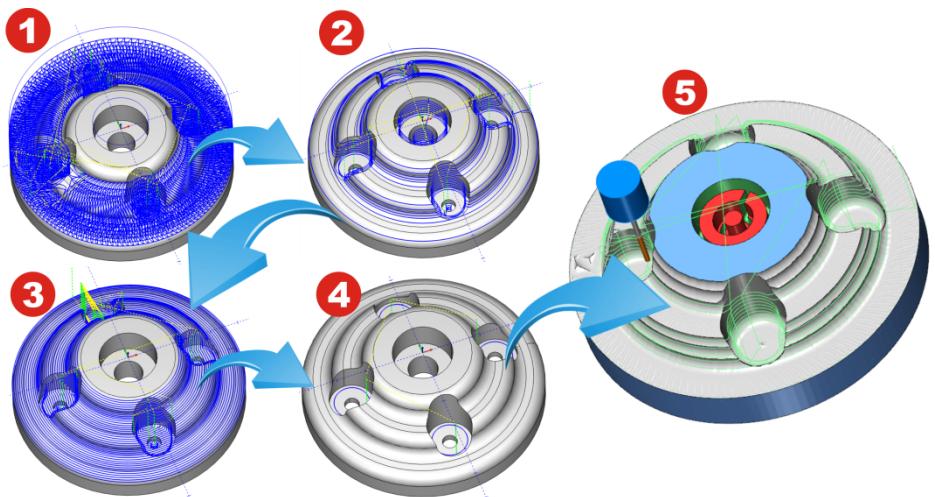


CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™



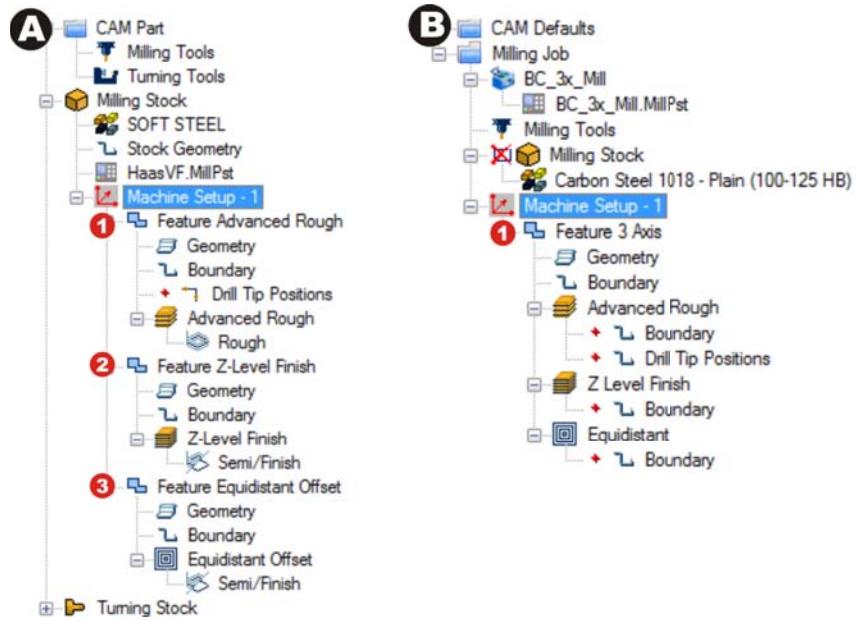
From the CAM Tree, all four operations are loaded into one strategy using DMS functionality; Dynamic Machining Strategies. Each machining operation toolpath can be contained within a boundary if required, however it is unnecessary to repetitively select geometry for all of the chosen machining operations.

This 3 Axis example has 1 strategy (3 Axis Milling) containing 4 machining operations. This is the heart of DMS. Then again, as in the case with 2 Axis machining, Wizards are used to step the user through the process of configuring what machining operations will be used in the strategy, and then inputting the data for each operation so that the output toolpath is right. Below you can see the CNC machining stages 1 through 4 and simulation 5.



Here is a different example. You can see the 2 CAM Trees (A and B) that have the same machining operations loaded into them.

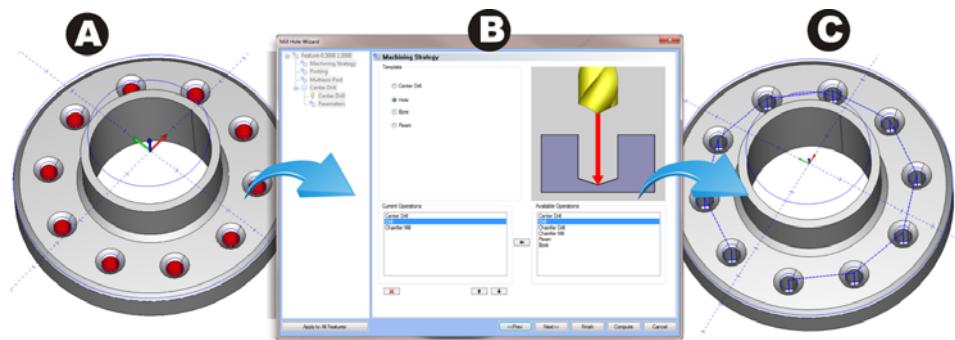
CAM Tree "A" shows 3 individually loaded operations into it. CAM Tree "B" shows one strategy that has 3 machining operations loaded under it. Both CAM Trees provide machining strategies and operations that are fully editable. Yet in CAM Tree "A" there are 3 sets of operations, 3 sets of geometry to select and 3 individual machining wizards to go through.



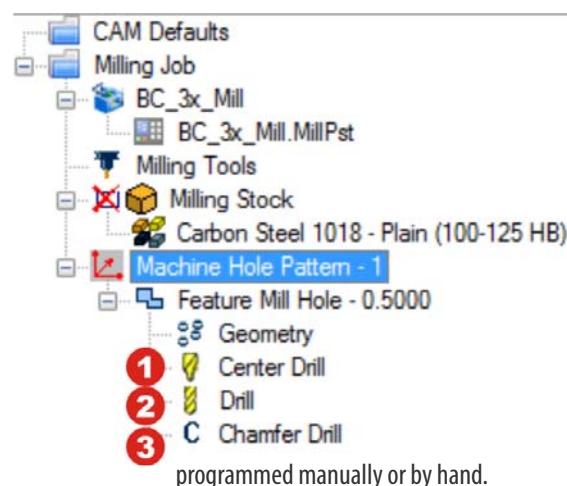


CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™

CAM Tree "B" is super efficient and all 3 machining operations are loaded into 1 machining strategy wizard with only one geometry selection needed to associate the operations to the part. This is a smarter and faster method of CAM programming. CAM Tree "B" still offers the ability to apply specific toolpaths to designated boundaries of the part as needed. CAM Tree "A" also shows unnecessary items such as Lathe when the job is a Mill part. That can be a little confusing. Dynamic Machining Strategies can also be used in Hole Making. CAD-CAM software may provide Multi-Tool Drilling operations in the form of Tool Patterns as well as through DMS within a CAM Machining Wizard exactly as used for the milling examples. Here is what that type of scenario would look like.



In "A" the drill holes are selected and the machining wizard is launched. In "B" the wizard provides the Dynamic Machining Strategies where a Center Drill, Drill and Chamfer operation is loaded into a "Hole" Strategy. Lastly, "C" shows the toolpath generated to machine the hole pattern. From this point the G-Code program is created and the job is simulated to ensure the accuracy of the job. Finally the NC program is sent to the machine to execute the program.

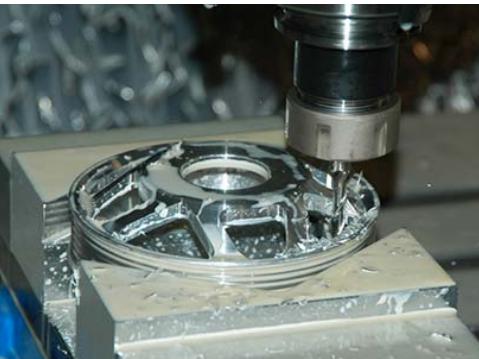


DMS functionality allows the programmer to load 3 operations into 1 strategy to save time and ensure that the machining wizard steps the programmer through each phase of the multi-tool drilling strategy successfully without wasting time or opening things up to a programming error due to something critical being skipped. This supports the level of workflow efficiency and accuracy that programmers need to machine their parts faster, smarter and easier than if programmed manually or by hand.

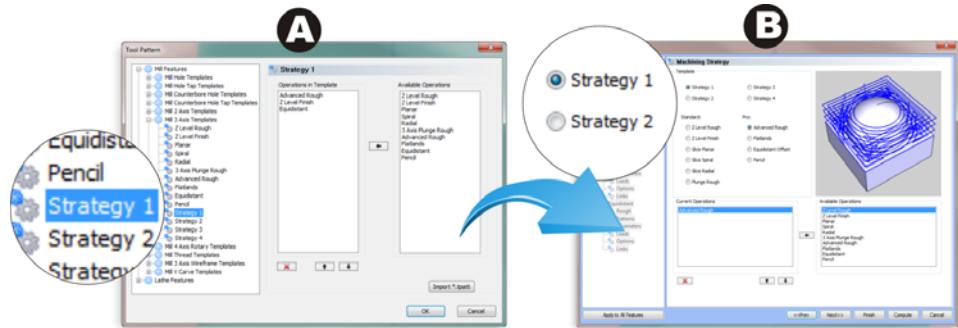
This CAM Tree displays 1 strategy (Feature-Mill Hole) with 3 machining operations under it. Only one wizard was required to program this multi-tool drilling situation. Once again, DMS proves to be a faster and more efficient programming method.



CAD-CAM Software | Wizard Driven
Dynamic Machining Strategies™



DMS functionality is also used in conjunction with what are called Tool Patterns "A". The user simply picks a machining strategy/feature to customize. Then the available operations can be chosen and added to the operations template easily as well as rearranged through using up and down arrow type buttons.



The tool pattern feature allows a programmer to choose any machining operation available in the software for a particular strategy. Later on this template will be made available in the appropriate CAM Wizard as a pre-setup strategy/template "B". Then, later on when a strategy template is chosen in the DMS functionality dialog of the wizard, the machining operations will automatically appear. Once the DMS page of the wizard is complete, all necessary steps of the wizard will be available and in one place for all loaded machining operations so that the guesswork is removed and the programmer can get on with machining parts faster. Again, eliminating the opportunity for something to be missed that relates to a operation and the resulting toolpath.

DMS provides improvement primarily for 3, 4 & 5 axis machining. However, you can see from these examples that DMS positively affects even 2 axis machining and hole drilling. In addition, you may add several finish passes simply by adding more operations to the machining feature. Operations do not share parameters, allowing independent control over each machining operation that is used. For example, you can have your finish operations start in a different location than your roughing operations. Ultimately, DMS provides CAM programmers more flexibility and control in employing machining operations for each strategy they use to cut CAD features.

BobCAD-CAM has provided CAD-CAM CNC Software products to the global manufacturing industry for over 25 years. The latest BobCAD-CAM software was used to show and explain Dynamic Machining Strategies in this paper as this software uses DMS functionality for machining. For more information on Dynamic Machining Strategies and CAD-CAM software call BobCAD-CAM, Inc. at 877-262-2231 or 727-442-3554. Visit www.bobcad.com for a free demo.

World Leader In Powerful & Affordable CAD/CAM Software

28200 U.S. Highway 19 N., Suite E

Clearwater, FL 33761



Toll Free: 877.262.2231

Intl: 727.442.3554

www.bobcad.com

